P515891.PDF [Page: 3 of 108]

Copy No. 18

DEFENCE R&D DÉFENSE

MATLAB COMPUTER PROGRAMS FOR VIBRATION AND CONTROL OF VIBRATIONS TRANSMITTED FROM MARINE ENGINES. USER MANUAL

S.G. Hutton

VIBE-TEK RESEARCH LTD.
5662 Holland Street
Vancouver
British Columbia
Canada
V6N 2A8

Defence R&D Canada

DEFENCE RESEARCH ESTABLISHMENT ATLANTIC

Contractor Report

DREA CR 2000-078

January 2001

National Defence Défense nationale

Canada

P515891.PDF [Page: 4 of 108]

REPRODUCTION QUALITY NOTICE

This document is the best quality available. The copy furnished to DRDCIM contained pages that may have the following quality problems:

: Pages smaller or Larger than normal

: Pages with background colour or light coloured printing

: Pages with small type or poor printing; and or

: Pages with continuous tone material or colour photographs

Due to various output media available these conditions may or may not cause poor legibility in the hardcopy output you receive.

☒ If this block is checked, the copy furnished to DRDCIM contained pages with colour printing, that when reproduced in Black and White, may change detail of the original copy.

Copy No: _____

MATLAB COMPUTER PROGRAMS FOR VIBRATION AND CONTROL OF VIBRATIONS TRANSMITTED FROM MARINE ENGINES. USERS MANUAL

S.G. Hutton
VIBE-TEK RESEARCH LTD.
5662 Holland Street
Vancouver, British Columbia, Canada, V6N 2A8

Contract No: W7707-9-7025/001/HAL and W7707-9-7293/001/HAL

The scientific or technical validity of this Contract Report is entirely the responsibility of the contractor and the contents do not necessarily have the approval or endorsement of Defence R&D Canada.

Defence Research Establishment Atlantic

Contractor Report

DREA CR 2000-078

January 2001

102-01725/

Contract Scientific Authorities

D.C. Stredulinsky J. P. Szabo N.C. Sponagle

Approved by

J.L. Kennedy

Head, Warship Signatures & Safety

Approved for release by

Kirk Foster

Chair/DRP

© Her Majesty the Queen as represented by the Minister of National Defence, 2001

© Sa majesté la reine, représentée par le ministre de la Défense nationale, 2001

Abstract

This report provides instructions for using the suite of MATLAB programs that have been developed in order to assist in the optimal vibration design of isolators for marine engines. These programs calculate the vibration transmissibility characteristics of the system and are of value in the design of optimal isolation systems. A program is also developed for the analysis of the active control of transmitted vibrations. The frequency range of interest is that encompassing the natural frequencies of the engine on its support system.

The general system that is analysed consists of an engine supported by vibration isolators on a foundation raft that itself is supported to ground by vibration isolators. The programs model the engine as a rigid body. The foundation raft can be modeled as a rigid body, or it can be modeled as a flexible body using beam finite elements. It is assumed that the isolators can be modeled as an isotropic linear viscoelastic material in terms of both a complex Young's modulus and a complex Poisson's Ratio, both of which are frequency dependant.

The report describes three different approaches to the modeling of the isolators: i) an approach based upon finite element analysis of the viscoelastic isolators using constitutive data of the isolator material; ii) an approach based upon the determination of the constitutive data from the results of dynamic stiffness measurements conducted on engine mounts; and iii) an approach based upon the results of so called 4-pole parameter data that describes the velocity force transfer matrices measured on full sized mounts in a given direction. In both the cases that involve experimentally determined isolator characteristics the data measured is not likely to completely describe the isolator characteristics. In such cases the programs developed have utilized an analytical approach to provide the missing data. In this way improved modeling of the actual structure is likely to occur. Numerical results are presented to illustrate each of the approaches developed.

A numerical problem that can occur in the finite element analysis of materials that are highly incompressible (as is the case with elastomeric materials used in vibration isolators) is described. A formulation is presented that avoids this problem. The report also contains the development of an active control option which is used to evaluate the potential for active control to be used in order to minimise the levels of force transmitted from the engine to the support.

Résumé

Le présent rapport fournit des instructions sur l'utilisation de la suite de programmes MATLAB qui ont été élaborés dans le but d'aider à la conception optimale des isolateurs de vibrations pour moteurs marins. Ces programmes établissent les caractéristiques de transmissibilité des vibrations du système et ils sont précieux pour la conception de systèmes d'isolation optimaux. Un programme est aussi élaboré pour l'analyse de la réduction active des vibrations transmises. La plage de fréquences d'intérêt est celle qui englobe les fréquences naturelles du moteur sur son support.

Le système général qui est analysé consiste en un moteur reposant sur des isolateurs de vibrations montés sur un radier, lequel prend appui au sol par l'intermédiaire d'autres isolateurs de vibrations. Les programmes modélisent le moteur comme étant un corps rigide. Le radier peut être modélisé comme un corps rigide, ou comme un corps flexible à l'aide d'éléments finis de poutre. On suppose que les isolateurs peuvent être modélisés comme un matériau viscoélastique linéaire isotrope en considérant un module de Young complexe et un coefficient de Poisson complexe, qui dépendent tous deux de la fréquence.

Le rapport décrit trois méthodes différentes de modélisation des isolateurs : i) une méthode basée sur l'analyse par éléments finis des isolateurs viscoélastiques faisant appel à des données constitutives du matériau de l'isolateur; ii) une méthode basée sur la détermination des données constitutives à partir des mesures de rigidité dynamique effectuées sur des supports de moteurs; iii) une méthode basée sur les données paramétriques dites quadripolaires qui décrivent les matrices de conversion de la vitesse en force établies pour des supports en grandeur réelle dans une direction donnée. Dans les deux méthodes basées sur les caractéristiques des isolateurs déterminées de façon expérimentale, les données obtenues ne semblent pas décrire complètement les caractéristiques des isolateurs. On a alors utilisé, dans les programmes élaborés, une méthode analytique permettant d'obtenir les données manquantes. On a ainsi des chances d'améliorer la modélisation de la structure réelle. Des résultats numériques sont présentés pour illustrer chacune des méthodes élaborées.

On décrit un problème numérique qui peut se poser dans l'analyse par éléments finis de matériaux hautement incompressibles (par exemple les élastomères utilisés dans les isolateurs de vibrations). On présente une façon de contourner ce problème. Le rapport traite aussi de l'élaboration d'une option de réduction active utilisée pour évaluer les possibilités de réduction active en vue de minimiser les niveaux de la force transmise du moteur au support.

Executive summary

Background

Marine diesel engines are supported by mounts that are designed to provide both structural rigidity and vibration isolation. Structural rigidity is required in order to maintain alignment of connecting shafts and piping; and vibration isolation is required in order to minimize the vibrations generated by the engine from being transmitted to the rest of the vessel and beyond. The vibration isolation materials currently in service on Canadian vessels, natural rubber and chloroprene rubber, experience large changes in their mechanical properties when exposed to lubricating oil or fuel. Replacing these materials with alternative elastomers could prevent alignment problems due to mount degradation, as well as reduce maintenance costs by increasing the length of time between mount changes. Replacement mounts must have a higher resistance to hydrocarbons and they must provide optimal vibration isolation. The optimal design of such mounts from a vibration viewpoint is complicated by the complex constitutive characteristics of the elastomers used in the construction of the mounts, and also by the complex nature of the three-dimensional vibration motion of the engine.

Principal Results

The principal result arising from this work is the further development of a computer program that enables the full three-dimensional vibration behaviour of a diesel engine mounted on polymeric isolators to be analyzed. The modeling capability of the program with respect to the engine support system has been extended. The program now has the ability to model an engine supported by an independent foundation raft, which is itself supported by ground. The isolators can be modeled on the basis of the geometric and constitutive properties of the isolator or they can be modeled on the basis of test results conducted on full-scale isolators. The program's capability has been extended to enable the modeling of isolators that have both complex Young's Modulus and complex Poisson's Ratio. An active control program has been implemented as an integral part of the VVES program and this will be used to investigate the possibilities of implementing active control of the engine transmitted vibrations.

Significance of Results

i) A sophisticated computational tool has been developed that will be of value in the optimal design of new isolators for marine diesel engines.

ii) An active control program has been developed that will assist in the evaluation of the future role of active control of engine vibrations.

Hutton, S.G. 2000. MATLAB Computer Programs for Vibration and Control of Vibrations Transmitted from Marine Engines: Users Manual. CR 2000-078 Defence Research Establishment Atlantic.

Sommaire

Renseignements généraux

Les moteurs diesel marins ont des supports qui sont conçus pour assurer la rigidité structurale et un amortissement des vibrations. La rigidité structurale est essentielle au maintien de l'alignement des arbres et de la tuyauterie. L'amortissement des vibrations est requise pour réduire au minimum la transmission des vibrations produites par le moteur au reste du navire et au-delà. L'huile de graissage et les carburants influent beaucoup sur les propriétés mécaniques des matériaux utilisés actuellement à bord des navires canadiens pour amortir les vibrations, le caoutchouc naturel et le polychloroprène. Si ces matériaux étaient remplacés par d'autres élastomères, cela pourrait prévenir les problèmes d'alignement causés par la détérioration des supports et réduire les coûts d'entretien en réduisant la fréquence de remplacement des supports. Les supports de rechange doivent avoir une meilleure résistance aux hydrocarbures et doivent fournir un amortissement optimal des vibrations. La conception de ces supports, du point de vue des vibrations, est compliquée par les caractéristiques constitutives complexes des élastomères utilisés pour la fabrication des supports et par la nature complexe des mouvements vibratoires tridimensionnels du moteur.

Principaux résultats

Le principal résultat qui ressort de ces travaux est l'élaboration d'un programme informatique permettant d'analyser le comportement vibratoire tridimensionnel d'un moteur diesel placé sur des isolateurs en polymère. La capacité de modélisation du programme relativement au système de support de moteur a été augmentée. Le programme peut maintenant modéliser un moteur supporté par un radier indépendant qui est fixé à la structure du navire. Les isolateurs peuvent être modélisés en fonction des propriétés géométriques et constitutives de l'isolateur ou peuvent être modélisés en fonction des résultats des essais effectués sur des isolateurs de grandeur réelle. La capacité du programme a été augmentée pour permettre la modélisation d'isolateurs ayant un module de Young complexe et un coefficient de Poisson complexe. Un programme de contrôle actif a été intégré au programme VVES et sera utilisé pour étudier la possibilité d'appliquer un contrôle actif des vibrations transmises par le moteur.

Importance des résultats

i) Un outil de calcul complexe a été mis au point pour aider à la conception optimale de nouveaux isolateurs pour les moteurs diesel marins.

ii) Un programme de contrôle actif a été élaboré afin d'aider à l'évaluation du rôle futur du contrôle actif sur les vibrations de moteur.

Hutton, S.G. 2000. Programmes Informatiques MATLAB pour l'Étude des Vibrations et la Réduction des Vibrations Transmises à Partir des Moteurs Marins: Manuel de l'Utilisateur. CR 2000-077 Defence Research Establishment Atlantic.

TABLE OF CONTENTS

	Page
Abstract	(i)
Résumé	(ii)
Executive Summary	(iii)
French Executive Summary	(iv)
Table of Contents	(v)
1. Introduction	1
2. Details of Programs	4
2.1 Calculation of System Matrices	4
2.1.1 VVESANALYSIS.m	4
2.1.2 VVESDYNSTFF.m	4
2.1.3 VVES4POLE.m	5
2.1.4 VVESACTIVE.m	7
2.2 Analysis of Free Vibration	8
2.2.1 VVESANALYSIS.m	8
2.2.2 VVESDYNSTFF.m	9
2.2.3 VVES4POLE.m	10
2.2.4 VVESACTIVE.m	10
2.3 Analysis of Frequency Response Functions	12
2.3.1 VVESANALYSIS.m	12
2.3.2 VVESDYNSTFF.m	12
2.3.3 VVESACTIVE.m	13
2.4 Analysis of Force Transmissibility Functions	17
2.4.1 VVESANALYSIS.m	17
2.4.2 VVESDYNSTFF.m	18
2.4.3 VVESACTIVE.m	19
2.5 Analysis of Displacement Response	21
2.5.1 VVESANALYSIS.m	21
2.5.2 VVESDYNSTFF.m	22
2.5.3 VVESACTIVE.m	23
2.6 Quasi-Static Solution to Roll and Pitch Motion	26
2.6.1 VVESANALYSIS.m	26
2.6.2 VVESDYNSTFF.m	26
2.6.3 VVESACTIVE.m	27
2.7 Active Control	29
3. Details of input files	35
3.1 Modeling of the engine	35
3.2 Modeling of the foundation raft	36
3.2.1 Modeling of the raft as a rigid body	36
3.2.2 Modeling of the raft as a flexible body	36

TABLE OF CONTENTS (continued)

3.3 Modeling of the isolators 38
3.4 Modeling the geometry of the complete system 45
3.5 Definition of the type of forced response analysis to be conducted 48
3.5.1 Frequency Response 48
3.5.2. Force Transmissibility 49
3.5.2.1 Force Transmissibility for Given Isolator 49
3.5.2.2 Total Force Transmissibility 49
3.5.3 Displacement Transmissibility 50
3.5.4 Active Control 50

4. Examples 54
4.1 VVESANALYSIS.m -Engine Supported on a Rigid Raft 54
4.2 VVESDYNSTFF.m -Engine Supported on a Rigid Raft. 68
4.3 VVES4POLE.m -Engine Supported on a Rigid Raft. 74
4.4 VVESACTIVE -Engine Supported on a Rigid Raft. 84

5. Appendix – VVESACTIVE Simulink Block Codes 92

1. Introduction

These MATLAB programs were written primarily for the analysis of the vibration and transmissibility characteristics of a marine engine supported by viscoelastic isolators when subjected to harmonic excitation. This manual describes how to run the different programs that have been developed.

In these programs the engine is assumed to be supported on a foundation raft by viscoelastic isolators and the raft itself is connected to ground by a second layer of isolators.

There are two options for modeling the raft:

a) the raft is modeled as a rigid body with the appropriate inertia characteristics provided by the user;
b) the raft is modeled in terms of finite element beam elements.

It is assumed that the isolators are composed of viscoelastic material with the characteristic that for harmonic response both the complex Young's Modulus and complex Poisson's ratio have both real and imaginary components.

The modeling assumes that the engine responds as a rigid body and thus the upper surface of the supporting isolators is constrained to lie on a plane surface. Similar assumptions are made for the rigid raft option. In the flexible raft option the isolators are connected to the centre line of the beams used to model the raft and are assumed to have the same displacements and rotations as the beam at the connection point. Each isolator can have independent geometric and constitutive characteristics. The isolators are represented by condensed 12x12 mass and stiffness matrices that can be determined by a number of different procedures depending upon the data available. The calculations conducted in the program are based upon a condensed finite element idealization of the isolator using three-dimensional solid elements. The global coordinate system used in this program is:

X-direction - longitudinal axis of the ship
Y-direction - lateral direction
Z-direction - vertical direction

Thus rotation about the X-axis (Rx) is roll and rotation about the Y-axis (Ry) is pitch.

Local x, y, z coordinates are used to describe the properties of the elastic beam elements, which are used to model the flexible support structure. In this case the z direction is the axial direction and the y direction is the vertical direction.

In the case of the isolators the local z coordinate is normal to the surface of the isolator pad.

The following options are available:

- VVESANALYSIS.m

  In this program the characteristics of the isolators are determined by analysis. This analysis is based upon the geometry and constitutive properties of the isolators. The constitutive parameters are provided by the user in order to describe the isotropic viscoelastic characteristics of the isolators.

  An automated finite element analysis is conducted for either rectangular block or tapered cylindrical shaped isolators. The program then condenses the finite element analysis into 12x12 mass and stiffness matrices which are used in the response

analysis of the total engine-raft-isolator system.
The condensed mass and stiffness matrices for the isolators can be provided directly to the program if they are known.

- VVESDYNSTFF.m
  In this program it is assumed that experimental data is available from measurements of the dynamic stiffness of the isolator. It is assumed that measurements are available in the x, y and z directions (corresponding to the appropriate diagonal components of the condensed isolator dynamic stiffness matrix). These measurements are used in the program to estimated the constitutive characteristics of the isolators, and then the same analysis procedure as in VVESANALYSIS.m is used in order to determine the determine the response characteristics of the system

- VVES4POLE.m
  In this program it is assumed that experiments have been conducted in order to determine the four-pole parameters matrices for the isolator in the x, y and z directions. Such matrices are the unidirectional transfer matrices involving velocity and force and in this case the solution procedure follows a transfer matrix approach and thus its formulation is basically different to the other programs. Three options are availabie.

  Case 1: In this case the complete 12x12 transmissibility parameter matrices for isolator layer 1 and 2 are produced by the program based upon analysis of the geometric and constitutive properties of the isolators.

  Case 2: This case uses only the experimentally determined 4-pole parameter matrices in the x, y and z directions. Thus in this case the transmissibility parameter matrices are not fully defined.

  Case 3: This case is the same as case 2 but with the unmeasured components of the incomplete transmissibility matrix being determined as in case 1.
  . This information is then used directly in the program with the unmeasured characteristics being determined using the same data as used in VVESANALYSIS.m

- VVESACTIVE.m
  This program is an extension of VVESANALYSIS.m in which an active control algorithm is implemented in order to reduce the force transmissibility of the system with respect to machinery induced forces that are transmitted to the foundation in the frequency range d.c. to 50Hz.

Table 1 lists the programs' capabilities.

Program Capability	vvesanalysis	vvesdynstff	vves4pole	vvesactive
Natural Frequencies, Mode Shapes	X	X		X
Frequency Response Functions	X	X	X	X
Force Transmissibility Functions	X	X	X	X
Prescribed Base Motion Response	X	X	X	
Displacement Due to Gravity Load	X	X		X
Active Control Algorithm				X
Rigid Raft	X	X	X	X
Flexible Raft	X	X		

Table 1 Summary of Program Capabilities

2. Details of the Programs

2.1 Calculation of System Matrices:

2.1.1 VVESANALYSIS.m

Program VVESANALYSIS.m calculates the system elastic stiffness matrix, the system viscoelastic stiffness matrix and the system mass matrix. These matrices are then used in subsequent analyses. The program requests the user to input a scale factor for the geometric configuration:

"Please input the scale factor for the geometry
The recommended value is 1: scale1= "

It also asks:

" If this is the first run of this example
or if visco or coord or nxyz have changed input 1, otherwise, input 2"

Input data files (not all may be required for any particular run):

COORD.ext	NXYZ.ext	RAFT.ext
VISCO.ext	DVISCO.ext	
KVE.ext	MVE.ext	
NODE.ext	ELEMENT.ext	
CONSTR.ext	ENGINE.ext	
BEAMS.ext	MASSES.ext	

Output figure:

Figure (1): Geometry of the undeformed system

The engine is represented by the colour black, the isolators by the colour red, raft and the elastic beam elements by blue and the foundation raft by yellow

Output file:

VVESANALYSIS.INP: The file VVESANALYSIS.INP contains a summary of all the input data for the problem. This file is created by the program and may be accessed from the directory that the problem is running from. Note that this file must be closed when the program is running or the program will crash.

2.1.2 VVESDYNSTFF.m

Program VVESDYNSTFF.m calculates the system elastic stiffness matrix, the system viscoelastic stiffness matrix and the system mass matrix. These matrices are then used in

subsequent analyses. The program requests the user to input a scale factor for the geometric configuration:

"Please input the scale factor for the geometry
The recommended value is 1: scale1= "

It also asks:

"If this is the first run of this example
or if visco or coord or nxyz have changed input 1, otherwise, input 2"

Input data files (not all may be required for any particular run):

COORD.ext	NXYZ.ext	RAFT.ext
VISCODS.ext	DIRECTSTIFF.ext	
NODE.ext	ELEMENT.ext	
CONSTR.ext	ENGINE.ext	
BEAMS.ext	MASSES.ext	

Output figure:

Figure (1): Geometry of the undeformed system

The engine is represented by the colour black, the isolators by the colour red, raft and the elastic beam elements by blue and the foundation raft by yellow

Output file: VVESDYNSTFF.INP:

The file VVESDYNSTFF.INP contains a summary of all the input data for the problem. This file is created by the program and may be accessed from the directory that the problem is running from. Note that this file must be closed when the program is running or the program will crash.

2.1.3 VVES4POLE.m

Program VVES4POLE.m calculates the force transmissibility, the velocity transmissibility, and the frequency response functions using a transfer function matrix approach. This program enables the input of measured four-pole data, and the computation of the response based only upon the measured data or based upon the measured data supplemented by calculated data. The program requests the user to input the case number:

"Input the case parameter. ncase=1 for computed transmissibility matrix input
ncase=2 for experimental 4-pole parameter matrix input
ncase=3 for experimental 4-pole parameter matrix input plus computed data
ncase = "

Case 1: In this case the complete 12x12 transmissibility parameter matrices for isolator layer 1 and 2 are produced by the program . based upon analysis of the geometric and constutive properties of the isolators The following data files are required :

COORD.ext	NXYZ.ext	VISCO.ext	NFREQ.ext
ENGINE.ext	ELEMENT4.ext	NODE.ext	RAFT.ext

Case 2: This case uses only the experimentally determined 4-pole parameter matrices directly to form incomplete transmissibility parameter matrices. Six 4-pole parameter matrices are needed for this case, three corresponding to the top layer and three corresponding to the bottom layer of isolators. The required input data files are:

FPX1.ext	FPY1.ext	FPZ1.ext	NODE.ext
FPX2.ext	FPY2.ext	FPZ2.ext	RAFT.ext
OFHZ.ext	ENGINE.ext	ELEMENT4.ext	

Case 3: This case is the same as case 2 but with the unmeasured components of the incomplete transmissibility matrix being determined as in case 1. The required input data files are:

COORD.ext	NXYZ.ext	VISCO.ext	NODE.ext
FPX1.ext	FPY1.ext	FPZ1.ext	RAFT.ext
FPX2.ext	FPY2.ext	FPZ2.ext	
OFHZ.ext	ENGINE.ext	ELEMENT4.ext	

Output figures:

Figure (71): Geometry of the undeformed system

The engine is represented by the colour black, the isolators by the colour red, raft and the elastic beam elements by blue and the foundation raft by yellow

Figure (72): Diagonal Components of the Force Transmissibility Matrix
Figure (73): Diagonal Components of the Base Velocity Transmissibility Matrix
Figure (74): Diagonal Components of the Engine Frequency Response Function Matrix

The output results will be contained in three matrices:
TTf: total force transmissibility functions (1:6,1:6, 1:MNUM) between the centre of mass of the engine and the prescribed base point
TTv: velocity transmissibility functions (1:6, 1:6, 1:MNUM) between the prescribed base point and the centre of mass of the engine
Hr: velocity frequency response functions (1:6, 1:6, 1:MNUM) where excitation and response are measured at the centre of mass of the engine.

The degrees of freedom in the above matrices correspond to forces in the X,Y,and

Z directions, the corresponding moments, and the associated velocities.

2.1.4 VVESACTIVE.m

Program VVESACTIVE.m calculates the system elastic stiffness matrix, the system viscoelastic stiffness matrix and the system mass matrix. These matrices are then used in subsequent analyses. The program requests the user to input a scale factor for the geometric configuration:

"Please input the scale factor for the geometry
The recommended value is 1: scale1="

It also asks:

" If this is the first run of this example
or if visco or coord or nxyz have changed input 1, otherwise, input 2"

Input data files (not all may be required for any particular run):

COORD.ext	NXYZ.ext	RAFT.ext
VISCO.ext	DVISCO.ext	
KVE.ext	MVE.ext	
NODE.ext	ELEMENT.ext	
CONSTR.ext	ENGINE.ext	
BEAMS.ext	MASSES.ext	

Output figure:

Figure (1): Geometry of the undeformed system

The engine is represented by the colour black, the isolators by the colour red, raft and the elastic beam elements by blue and the foundation raft by yellow

Output file:

VVESACTIVE.INP: The file VVESACTIVE.INP contains a summary of all the input data for the problem. This file is created by the program and may be accessed from the directory that the problem is running from. Note that this file must be closed when the program is running or the program will crash.

2.2 Analysis of Free Vibration

2.2.1 VVESANALYSIS

The analysis of free vibration has two options: one if the whole system is composed entirely of elastic elements, in which all the eigenvalues and eigenvectors are real; or secondly if the isolators are viscoelastic, in which case the eigenvalues and eigenvectors are complex.

Program EIGENANALYSIS.m calculates the eigenvalues and eigenvectors of the elastic problem and requests the user to input the number of eingenvalues (natural frequencies) to be solved for:

"Please input the number of eigenvalues to be solved, NFREQ ="

Output files:

VVESANALYSIS.1 natural frequencies (Hz)

Program VEIGENANALYSIS.m calculates the eigenvalues and eigenvectors of the viscoelastic problem and requests the user to input the number of eingenvalues (natural frequencies) to be solved for:

"Please input the number of eigenvalues to be solved, NFREQ ="

Output files:

VVESANALYSIS.2 complex eigenvalues (Hz)

Program MODE.m plots the mode shapes for both EIGENANALYSIS and VEIGENANALYSIS. The program requests the user to input the scale factor for modal deformation

"Please input the scale factor for modal deformation
The recommended value is 1: Scale2="

Output figures:

Figure (11): First mode shape.

.....

Figure (10+NFREQ): Last mode shape.

Program ANIM.m animates the modal shapes. The user is prompted for the required mode and a scale factor .

" Please input the mode number: Ir ="

"Please input the scale factor. The recommended value is 1: Scale="

The animation will stop after 15 seconds, if further viewing of this mode is required enter "1" if another mode is required enter "0" and run ANIM.m again.

Output figures:

Figure (2): Animation of the Ir mode shape.

A record of the calculated eigenvectors and eigenvalues can be found in MATLAB in the matrices NATFREQ and MODES. The data contained in NATFREQ consists of two numbers. The first is the decay coefficient (Hz), the more negative this number the greater degree of damping that exists in this mode. The second number is the natural frequency (Hz).

2.2.2 VVESDYNSTFF

The analysis of free vibration has two options: one if the whole system is composed entirely of elastic elements, in which all the eigenvalues and eigenvectors are real; or secondly if the isolators are viscoelastic, in which case the eigenvalues and eigenvectors are complex.

Program EIGENDYNSTFF.m calculates the eigenvalues and eigenvectors of the elastic problem and requests the user to input the number of eingenvalues (natural frequencies) to be solved for:

"Please input the number of eigenvalues to be solved, NFREQ ="

Output files: VVESDYNSTFF.1 natural frequencies (Hz)

Program VEIGENDYNSTFF.m calculates the eigenvalues and eigenvectors of the viscoelastic problem and requests the user to input the number of eingenvalues (natural frequencies) to be solved for:

"Please input the number of eigenvalues to be solved, NFREQ ="

Output files:

VVESDYNSTFF.2 complex eigenvalues (Hz)

Program MODE.m plots the mode shapes for both EIGENDYNSTFF and VEIGENDYNSTFF. The program requests the user to input the scale factor for modal deformation

"Please input the scale factor for modal deformation
The recommended value is 1: Scale2="

Output figures:

Figure (11): First mode shape.

.....

Figure(10+NFREQ): Last mode shape.

Program ANIM.m animates the modal shapes. The user is prompted for the required mode and a scale factor.

" Please input the mode number: Ir ="

"Please input the scale factor. The recommended value is 1: Scale="

The animation will stop after 15 seconds, if further viewing of this mode is required enter "1" if another mode is required enter "0" and run ANIM.m again.

Output figures:

Figure (2): Animation of the Ir mode shape.

A record of the calculated eigenvectors and eigenvalues can be found in MATLAB in the matrices NATFREQ and MODES. The data contained in NATFREQ consists of two numbers. The first is the decay coefficient (Hz), the more negative this number the greater degree of damping that exists in this mode. The second number is the natural frequency (Hz).

2.2.3 VVES4POLE

Natural frequency analysis is not available for VVES4POLE.

2.2.4 VVESACTIVE

The analysis of free vibration has two options: one if the whole system is composed entirely of elastic elements, in which all the eigenvalues and eigenvectors are real; or secondly if the isolators are viscoelastic, in which case the eigenvalues and eigenvectors are complex.

Program EIGENACTIVE.m calculates the eigenvalues and eigenvectors of the elastic problem and requests the user to input the number of eingenvalues (natural frequencies) to be solved for:

"Please input the number of eigenvalues to be solved, NFREQ ="

Output files:

VVESACTIVE.1 natural frequencies (Hz)

Program VEIGENACTIVE.m calculates the eigenvalues and eigenvectors of the viscoelastic problem and requests the user to input the number of eingenvalues (natural frequencies) to be solved for:

"Please input the number of eigenvalues to be solved, NFREQ ="

Output files:

VVESACTIVE.2 complex eigenvalues (Hz)

Program MODE.m plots the mode shapes for both EIGENACTIVE and VEIGENACTIVE. The program requests the user to input the scale factor for modal deformation

"Please input the scale factor for modal deformation
The recommended value is 1: Scale2="

Output figures:

Figure (11): First mode shape.

⋮

Figure (10+NFREQ): Last mode shape.

Program ANIM.m animates the modal shapes. The user is prompted for the required mode and a scale factor (.1 is generally appropriate).

" Please input the mode number: Ir ="

"Please input the scale factor. The recommended value is 1: Scale="

The animation will stop after 15 seconds, if further viewing of this mode is required enter "1" if another mode is required enter "0" and run ANIM.m again.

Output figures:

Figure (2): Animation of the Ir mode shape.

A record of the calculated eigenvectors and eigenvalues can be found in MATLAB in the matrices NATFREQ and MODES. The data contained in NATFREQ consists of two numbers. The first is the decay coefficient (Hz), the more negative this number the greater degree of damping that exists in this mode. The second number is the natural frequency (Hz).

2.3 Analysis of Frequency Response Functions

2.3.1 VVESANALYSIS

Program FRFUNANALYSIS.m calculates frequency response functions (magnitude and phase) between two prescribed nodes. At the first node a harmonically varying unit force or moment is applied and at the second node the response is calculated for all six degrees of freedom. The displacement, velocity, and acceleration responses are calculated. The graphical and stored output units are (response units/input units).

Input data files:

FORCE.ext
RESP.ext

Output files:

VVESANALYSIS.3 Acceleration frequency response functions
VVESANALYSIS.4 Velocity frequency response functions
VVESANALYSIS.5 Displacement frequency response functions

Output file format:

First line: [excited node, excited degree of freedom]
Second line: [response node]
Subsequent lines: [frequency (Hz); Tx; Ty; Tz; Rx; Ry; Rz]
where Tx is the lateral response in the x direction and Rx is the rotational response about the x axis.

Output figures:

Figure (3): Magnitude of velocity frequency response functions, linear scale
Figure (4): Acceleration frequency response functions, magnitude and phase
Figure (5): Velocity frequency response functions, magnitude and phase
Figure (6): Displacement frequency response functions, magnitude and phase

2.3.2 VVESDYNSTFF

Program FRFUNDYNSTFF.m calculates frequency response functions (magnitude and phase) between two prescribed nodes. At the first node a harmonically varying unit force or moment is applied and at the second node the response is calculated for all six degrees of freedom. The displacement, velocity, and acceleration responses are calculated. The graphical and stored output units are (response units/input units).

Input data files:

FORCE.ext

RESP.ext

Output files:

VVESDYNSTFF.3 Acceleration frequency response functions
VVESDYNSTFF.4 Velocity frequency response functions
VVESDYNSTFF.5 Displacement frequency response functions

Output file format:

First line: [excited node, excited degree of freedom]
Second line: [response node]
Subsequent lines: [frequency (Hz); Tx; Ty; Tz; Rx; Ry; Rz]
where Tx is the lateral response in the x direction and Rx is the rotational response about the x axis.

Output figures:

Figure (3): Magnitude of velocity frequency response functions, linear scale
Figure (4): Acceleration frequency response functions, magnitude and phase
Figure (5): Velocity frequency response functions, magnitude and phase
Figure (6): Displacement frequency response functions, magnitude and phase

2.3.3 VVESACTIVE

Program FRFUNACTIVE.m calculates frequency response functions (magnitude and phase) between two prescribed nodes. At the first node a harmonically varying unit force or moment is applied and at the second node the response is calculated for all six degrees of freedom. The displacement, velocity, and acceleration responses are calculated. The graphical and stored output units are (response units/input units).

Input data files:

FORCE.ext
RESP.ext

Output files:

VVESACTIVE.3 Acceleration frequency response functions
VVESACTIVE.4 Velocity frequency response functions
VVESACTIVE.5 Displacement frequency response functions

Output file format:

First line: [excited node, excited degree of freedom]
Second line: [response node]
Subsequent lines: [frequency (Hz); Tx; Ty; Tz; Rx; Ry; Rz]
where Tx is the lateral response in the x direction and Rx is the rotational response about the x axis.

Output figures:

Figure (3): Magnitude of velocity frequency response functions, linear scale
Figure (4): Acceleration frequency response functions, magnitude and phase
Figure (5): Velocity frequency response functions, magnitude and phase
Figure (6): Displacement frequency response functions, magnitude and phase

File name: FRFACTIVE.m

Purpose: to calculate the whole matrix (for all degrees of freedom) of displacement frequency response functions for the case where the raft is idealized as a rigid body.

function [omhz,frf]=frfactive(MINF,MAXF,MNUM,M22,K22,Kv22,visco,dvisco,NDV)

Output Matrices:

omhz: size(NFREQ,1) ----- frequencies at which the frequency response function is calculated.
frf : size(N, N, NFREQ) ----- frequency response function matrix where N = total number of engine and raft degrees of freedom. For rigid raft case, N=12.

In frf the degrees of freedom are as follows:
1,2,3: displacements of the engine centre of mass in the global X,Y, and Z directions;
4,5,6: rotations of the engine about the global X,Y,Z axes;
7,8,9: displacements of the raft centre of mass in the global X,Y, and Z directions;
10,11,12: rotations of the raft about the global X,Y,Z axes.

Input variables and matrices:

To be supplied:

MINF ---- minimum frequency (Hz)
MAXF ----- maximum frequency (Hz)
MNUM-------- number of frequency points at which frequency response matrix is calculated.

The following input variables are created by VVESACTIVE.m:

M22 ------ system mass matrix
K22 ------- system elastic stiffness matrix
Kv22 ------- system viscoelastic stiffness matrix
visco ------- parameter matrix of viscoelastic materials
dvisco ------ discrete parameter matrix of viscoelastic materials
NDV -------- an index to indicate if visco or dvisco is used

During calculation of the frequency response functions, only MINF, MAXF and MNUM need to be input, since all other variables and matrices are defined after running VVESACTIVE.m

File name: FRFSISO.m

Purpose: to calculate the frequency response function matrix between any two points on the engine or raft. It assumes that the raft is modeled as a rigid body. The frequency response functions are calculated with respect to forces and displacements in the direction of the global coordinate axes and moments and rotations about these axes.

function[omhz,frfsiso]
=frfsiso(NI,SI,NO,SO,MINF,MAXF,MNUM,M22,K22,Kv22,visco,dvisco,NDV,node,co
nstrn,DOF1)

Output Matrices:

omhz: size(NFREQ,1) ----- frequencies at which the frequency response functions are calculated.

frfsiso: size(6, 6, NFREQ) ----- frequency response function matrix

Input variables and matrices:

To be supplied:

NI----Node number of the center of mass of engine or rigid raft depending upon location of the load.

SI ---- (1,3) vector. The coordinates of the point at which the force is applied.

NO---- Node number of the center of mass of engine or rigid raft depending upon location of the response.

SO ---- (1,3) vector. The coordinates of the point at which the response is to be measured.

MINF ---- minimum frequency (Hz)

MAXF ----- maximum frequency (Hz)

MNUM-------- number of frequency points at which frequency response matrix is calculated.

The following input variables are created by VVESACTIVE.m:

M22 ------ system mass matrix

K22 ------- system elastic stiffness matrix

Kv22 ------- system viscoelastic stiffness matrix

visco ------- parameter matrix of viscoelastic materials

dvisco ------ discrete parameter matrix of viscoelastic materials

NDV -------- an index to indicate if visco or dvisco is used

node ------- coordinates of system nodes

constrn----- node freedom number matrix

DOF1------ number of base degrees of freedom.

During calculation, only NI, NO, SI, SO, MINF, MAXF and MNUM need to be input, since all other variables and matrices are defined after running VVESACTIVE.

2.4 Analysis of Force Transmissibility Functions

2.4.1 VVESANALYSIS

Program FTRSSANALSIS.m calculates force transmissibility functions (magnitude and phase) between a prescribed excited node (generally the centre of mass of the engine) and a prescribed support node in both the Y and Z directions. The user may also specify an arbitrary location on the engine for the applied load.

Input data files:

FTRANS.ext

Output files:

VVESANALSIS.6 Fx Force transmissibility functions
VVESANALSIS.7 Fy Force transmissibility functions
VVESANALSIS.8 Fz Force transmissibility functions

Output file format:

First line: [excited node, or coordinates of excited node]
Second line: [response node]
Subsequent lines: [frequency (Hz); Tx; Ty; Tz; Rx; Ry; Rz]
Where Tx is the lateral response in the X direction and
Rx is the rotational response about the X axis.

Output figures:

Figure(31): Magnitude and phase of force transmissibility functions (Fy, Fz). Excitation Fy = 1.
Figure(32): Magnitude and phase of force transmissibility functions (Fy, Fz). Excitation Fz = 1.

Program TFTRSSANALSIS.m calculates the total force transmissibility functions (magnitude and phase) between a prescribed excited node (generally the centre of mass of the engine) and all support nodes in the X, Y and Z directions. The user may also specify an arbitrary location on the engine for the applied load.

Input data files:

TFTRANS.ext

Output files:

VVESANALSIS.16 Fx Force transmissibility functions
VVESANALSIS.17 Fy Force transmissibility functions
VVESANALSIS.18 Fz Force transmissibility functions

Output file format:

First line: [excited node, or coordinates of excited node]
Subsequent lines: [frequency (Hz); Tx; Ty; Tz;]
Where Tx is the lateral response in the X direction.

Output figures:

Figure(35): Magnitude and phase of force transmissibility functions (Fx, Fy, Fz). Excitation Fx = 1.
Figure(36): Magnitude and phase of force transmissibility functions (Fx, Fy, Fz). Excitation Fy = 1.
Figure(37): Magnitude and phase of force transmissibility functions (Fx, Fy, Fz). Excitation Fz = 1.

2.4.2 VVESDYNSTFF

Program FTRSSDYNSTFF.m calculates force transmissibility functions (magnitude and phase) between a prescribed excited node (generally the centre of mass of the engine) and a prescribed support node in both the Y and Z directions. The user may also specify an arbitrary location on the engine for the applied load.

Input data files:

FTRANS.ext

Output files:

VVESDYNSTFF.6 Fx Force transmissibility functions
VVESDYNSTFF.7 Fy Force transmissibility functions
VVESDYNSTFF.8 Fz Force transmissibility functions

Output file format:

First line: [excited node, or coordinates of excited node]
Second line: [response node]
Subsequent lines: [frequency (Hz); Tx; Ty; Tz; Rx; Ry; Rz]
Where Tx is the lateral response in the X direction and Rx is the rotational response about the X axis.

Output figures:

Figure(31): Magnitude and phase of force transmissibility functions (Fy, Fz). Excitation Fy = 1.
Figure(32): Magnitude and phase of force transmissibility functions (Fy, Fz). Excitation Fz = 1.

Program TFTRSSDYNSTFF.m calculates the total force transmissibility functions (magnitude and phase) between a prescribed excited node (generally the centre of mass of the engine) and all support nodes in the X, Y and Z directions. The user may also specify an arbitrary location on the engine for the applied load.

Input data files:

TFTRANS.ext

Output files:

VVESDYNSTFF.16 Fx Force transmissibility functions
VVESDYNSTFF.17 Fy Force transmissibility functions
VVESDYNSTFF.18 Fz Force transmissibility functions

Output file format:

First line: [excited node, or coordinates of excited node]
Subsequent lines: [frequency (Hz); Tx; Ty; Tz;]
Where Tx is the lateral response in the X direction.

Output figures:

Figure(35): Magnitude and phase of force transmissibility functions (Fx, Fy, Fz). Excitation Fx = 1.
Figure(36): Magnitude and phase of force transmissibility functions (Fx, Fy, Fz). Excitation Fy = 1.
Figure(37): Magnitude and phase of force transmissibility functions (Fx, Fy, Fz). Excitation Fz = 1.

2.4.3 VVESACTIVE

Program FTRSSACTIVE.m calculates force transmissibility functions (magnitude and phase) between a prescribed excited node (generally the centre of mass of the engine) and a prescribed support node in both the Y and Z directions. The user may also specify an arbitrary location on the engine for the applied load.

Input data files:

FTRANS.ext

Output files:

VVESACTIVE.6 Fx Force transmissibility functions
VVESACTIVE.7 Fy Force transmissibility functions
VVESACTIVE.8 Fz Force transmissibility functions

Output file format:

First line: [excited node, or coordinates of excited node]
Second line: [response node]
Subsequent lines: [frequency (Hz); Tx; Ty; Tz; Rx; Ry; Rz]
Where Tx is the lateral response in the X direction and
Rx is the rotational response about the X axis.

Output figures:

Figure(31): Magnitude and phase of force transmissibility functions (Fy, Fz). Excitation Fy = 1.
Figure(32): Magnitude and phase of force transmissibility functions (Fy, Fz). Excitation Fz = 1.

Program TFTRSSACTIVE.m calculates the total force transmissibility functions (magnitude and phase) between a prescribed excited node (generally the centre of mass of the engine) and all support nodes in the X, Y and Z directions. The user may also specify an arbitrary location on the engine for the applied load.

Input data files:

TFTRANS.ext

Output files:

VVESACTIVE.16 Fx Force transmissibility functions
VVESACTIVE.17 Fy Force transmissibility functions
VVESACTIVE.18 Fz Force transmissibility functions

Output file format:

First line: [excited node, or coordinates of excited node]
Subsequent lines: [frequency (Hz); Tx; Ty; Tz;]
Where Tx is the lateral response in the X direction.

Output figures:

Figure(35): Magnitude and phase of force transmissibility functions (Fx, Fy, Fz). Excitation Fx = 1.
Figure(36): Magnitude and phase of force transmissibility functions (Fx, Fy, Fz). Excitation Fy = 1.
Figure(37): Magnitude and phase of force transmissibility functions (Fx, Fy, Fz). Excitation Fz = 1.

2.5 Analysis of Displacement Response

2.5.1 VVESANALYSIS

Program DTRSSANALYSIS.m calculates displacement transmissibility functions (magnitude and phase) at a prescribed node for a given rigid body motion of the supports. The origin of the rigid body motion may be specified by the user. This analysis simulates the effect of ship motion on the response of the engine.

Input data files:

DTRANS.ext

Output files:

VVESANALYSIS.9	Displacement transmissibility functions excited by Tx
VVESANALYSIS.10	Displacement transmissibility functions excited by Ty
VVESANALYSIS.11	Displacement transmissibility functions excited by Tz
VVESANALYSIS.12	Displacement transmissibility functions excited by Rx
VVESANALYSIS.13	Displacement transmissibility functions excited by Ry
VVESANALYSIS.14	Displacement transmissibility functions excited by Rz

Output format:

first line: coordinates of base excitation point (x,y,z)
second line: type of excitation, =1: displacement along x-direction
=2 : displacement along y-direction
=3 : displacement along z-direction
=4 : rotation about x-axis
=5 : rotation about y-axis
=6 : rotation about z-axis
third line: response node number
The subsequent lines are: [Frequency(Hz);Tx;Ty;Tz;Rx;Ry;Rz]

Output figures:

Figure (41): Displacement transmissibility functions (Ty, Tz, Rx, Ry) magnitude for load Ty=1, linear scale
Figure (42): Displacement transmissibility functions (Ty, Tz, Rx, Ry) magnitude for load Tz=1, linear scale
Figure (43): Displacement transmissibility functions (Ty, Tz, Rx, Ry) magnitude for load Rx=1, linear scale
Figure (44): Displacement transmissibility functions (Ty, Tz, Rx, Ry) magnitude for load Ry=1, linear scale
Figure (45): Displacement transmissibility functions (Ty, Tz) magnitude and phase for load Ty=1, logarithmic scale
Figure (46): Displacement transmissibility functions (Rx, Ry) magnitude and phase for load Ty=1, logarithmic scale

Figure (47): Displacement transmissibility functions (Ty, Tz)
magnitude and phase for load Tz=1, logarithmic scale
Figure (48): Displacement transmissibility functions (Rx, Ry)
magnitude and phase for load Tz=1, logarithmic scale
Figure (49): Displacement transmissibility functions (Ty, Tz)
magnitude and phase for load Rx=1, logarithmic scale
Figure (50): Displacement transmissibility functions (Rx, Ry)
magnitude and phase for load Rx=1, logarithmic scale
Figure (51): Displacement transmissibility functions (Ty, Tz)
magnitude and phase for load Ry=1, logarithmic scale
Figure (52): Displacement transmissibility functions (Rx, Ry)
magnitude and phase for load Ry=1, logarithmic scale

2.5.2 VVESDYNSTFF

Program DTRSSDYNSTFF.m calculates displacement transmissibility functions (magnitude and phase) at a prescribed node for a given rigid body motion of the supports. The origin of the rigid body motion may be specified by the user. This analysis simulates the effect of ship motion on the response of the engine.

Input data files:

DTRANS.ext

Output files:

VVESDYNSTFF.9	Displacement transmissibility functions, excited by Tx
VVESDYNSTFF.10	Displacement transmissibility functions excited by Ty
VVESDYNSTFF.11	Displacement transmissibility functions excited by Tz
VVESDYNSTFF.12	Displacement transmissibility functions excited by Rx
VVESDYNSTFF.13	Displacement transmissibility functions excited by Ry
VVESDYNSTFF.14	Displacement transmissibility functions excited by Rz

Output format:

first line: coordinates of base excitation point (x,y,z)
second line: type of excitation, =1: displacement along x-direction
=2 : displacement along y-direction
=3 : displacement along z-direction
=4 : rotation about x-axis
=5 : rotation about y-axis
=6 : rotation about z-axis
third line: response node number
The subsequent lines are: [Frequency(Hz);Tx;Ty;Tz;Rx;Ry;Rz]

Output figures:

Figure (41): Displacement transmissibility functions (Ty, Tz, Rx, Ry) magnitude for load Ty=1, linear scale
Figure (42): Displacement transmissibility functions (Ty, Tz, Rx, Ry) magnitude for load Tz=1, linear scale
Figure (43): Displacement transmissibility functions (Ty, Tz, Rx, Ry) magnitude for load Rx=1, linear scale
Figure (44): Displacement transmissibility functions (Ty, Tz, Rx, Ry) magnitude for load Ry=1, linear scale
Figure (45): Displacement transmissibility functions (Ty, Tz) magnitude and phase for load Ty=1, logarithmic scale
Figure (46): Displacement transmissibility functions (Rx, Ry) magnitude and phase for load Ty=1, logarithmic scale
Figure (47): Displacement transmissibility functions (Ty, Tz) magnitude and phase for load Tz=1, logarithmic scale
Figure (48): Displacement transmissibility functions (Rx, Ry) magnitude and phase for load Tz=1, logarithmic scale
Figure (49): Displacement transmissibility functions (Ty, Tz) magnitude and phase for load Rx=1, logarithmic scale
Figure (50): Displacement transmissibility functions (Rx, Ry) magnitude and phase for load Rx=1, logarithmic scale
Figure (51): Displacement transmissibility functions (Ty, Tz) magnitude and phase for load Ry=1, logarithmic scale
Figure (52): Displacement transmissibility functions (Rx, Ry) magnitude and phase for load Ry=1, logarithmic scale

2.5.3 VVESACTIVE

Program DTRSSACTIVE.m calculates displacement transmissibility functions (magnitude and phase) at a prescribed node for a given rigid body motion of the supports. The origin of the rigid body motion may be specified by the user. This analysis simulates the effect of ship motion on the response of the engine.

Input data files:

DTRANS.ext

Output files:

VVESACTIVE.9	Displacement transmissibility functions excited by Tx
VVESACTIVE.10	Displacement transmissibility functions excited by Ty
VVESACTIVE.11	Displacement transmissibility functions excited by Tz
VVESACTIVE.12	Displacement transmissibility functions excited by Rx
VVESACTIVE.13	Displacement transmissibility functions excited by Ry
VVESACTIVE.14	Displacement transmissibility functions excited by Rz

Output format:

first line: coordinates of base excitation point (x,y,z)
second line: type of excitation, =1: displacement along x-direction
=2 : displacement along y-direction
=3 : displacement along z-direction
=4 : rotation about x-axis
=5 : rotation about y-axis
=6 : rotation about z-axis
third line: response node number
The subsequent lines are: [Frequency(Hz);Tx;Ty;Tz;Rx;Ry;Rz]

Output figures:

Figure (41): Displacement transmissibility functions (Ty, Tz, Rx, Ry)
magnitude for load Ty=1, linear scale
Figure (42): Displacement transmissibility functions (Ty, Tz, Rx, Ry)
magnitude for load Tz=1, linear scale
Figure (43): Displacement transmissibility functions (Ty, Tz, Rx, Ry)
magnitude for load Rx=1, linear scale
Figure (44): Displacement transmissibility functions (Ty, Tz, Rx, Ry)
magnitude for load Ry=1, linear scale
Figure (45): Displacement transmissibility functions (Ty, Tz)
magnitude and phase for load Ty=1, logarithmic scale
Figure (46): Displacement transmissibility functions (Rx, Ry)
magnitude and phase for load Ty=1, logarithmic scale
Figure (47): Displacement transmissibility functions (Ty, Tz)
magnitude and phase for load Tz=1, logarithmic scale
Figure (48): Displacement transmissibility functions (Rx, Ry)
magnitude and phase for load Tz=1, logarithmic scale
Figure (49): Displacement transmissibility functions (Ty, Tz)
magnitude and phase for load Rx=1, logarithmic scale
Figure (50): Displacement transmissibility functions (Rx, Ry)
magnitude and phase for load Rx=1, logarithmic scale
Figure (51): Displacement transmissibility functions (Ty, Tz)
magnitude and phase for load Ry=1, logarithmic scale
Figure (52): Displacement transmissibility functions (Rx, Ry)
magnitude and phase for load Ry=1, logarithmic scale

File name: ERRACTIVE.m

Purpose: to calculate the displacement response function vector for all global degrees of freedom for a force vector "forces" applied to a system with frequency response function matrix "frf".

```
function err=erractive(omhz,frf,forces)
```

Output Matrix:

err: size(MNUM+1,N1) -------- displacement response of system in the frequency domain. N1 is dependent on the dimension of frf .

Input Matrices:

omhz: size(MNUM+1,1) ----- ------- discrete frequency points
frf : size(N1, N1, MNUM+1) ----- frequency response function matrix
forces: size(MNUM+1,N1) ---------- force matrix in frequency domain

For the case of a rigid raft N1=12.

2.6 Quasi-Static Solution to Roll and Pitch Motion

2.6.1 VVESANALYSIS

Program QUASIANALYSIS.m calculates the maximum displacements caused by the application of a static gravity load caused as a result of (i) a roll motion of ROLL degrees and (ii) a pitch motion of PITCH degrees.

QUASI.ext (1,2) Vector defining the magnitude of the roll and pitch motions (in degrees) for which the maximum engine displacements are calculated.

Column 1: ROLL
Column 2: PITCH

Output files:
VVESANALYSIS.15 Quasi-static displacements

Output format:
Line 1:
Column 1: Maximum displacements caused by ship roll
Column 2: Tx for centre of mass caused by ship roll
Column 3: Ty for centre of mass caused by ship roll
Column 4: Tz for centre of mass caused by ship roll
Column 5: Rx for centre of mass caused by ship roll (degrees)
Column 6: Ry for centre of mass caused by ship roll (degrees)
Column 7: Rz for centre of mass caused by ship roll (degrees)
Line 2:
Column 1: Maximum displacements caused by ship pitch
Column 2: Tx for centre of mass caused by ship pitch
Column 3: Ty for centre of mass caused by ship pitch
Column 4: Tz for centre of mass caused by ship pitch
Column 5: Rx for centre of mass caused by ship pitch (degrees)
Column 6: Ry for centre of mass caused by ship pitch (degrees)
Column 7: Rz for centre of mass caused by ship pitch (degrees)

2.6.2 VVESDYNSTFF

Program QUASIDYNSTFF.m calculates the maximum displacements caused by the application of a static gravity load caused as a result of (i) a roll motion of ROLL degrees and (ii) a pitch motion of PITCH degrees.

QUASI.ext (1,2) Vector defining the magnitude of the roll and pitch motions (in degrees) for which the maximum engine displacements are calculated.

Column 1: ROLL
Column 2: PITCH

Output files:

VVESDYNSTFF.15 Quasi-static displacements

Output format:

Line 1:
Column 1: Maximum displacements caused by ship roll
Column 2: Tx for centre of mass caused by ship roll
Column 3: Ty for centre of mass caused by ship roll
Column 4: Tz for centre of mass caused by ship roll
Column 5: Rx for centre of mass caused by ship roll (degrees)
Column 6: Ry for centre of mass caused by ship roll (degrees)
Column 7: Rz for centre of mass caused by ship roll (degrees)
Line 2:
Column 1: Maximum displacements caused by ship pitch
Column 2: Tx for centre of mass caused by ship pitch
Column 3: Ty for centre of mass caused by ship pitch
Column 4: Tz for centre of mass caused by ship pitch
Column 5: Rx for centre of mass caused by ship pitch (degrees)
Column 6: Ry for centre of mass caused by ship pitch (degrees)
Column 7: Rz for centre of mass caused by ship pitch (degrees)

2.6.3 VVESACTIVE

Program QUASIACTIVE.m calculates the maximum displacements caused by the application of a static gravity load caused as a result of (i) a roll motion of ROLL degrees and (ii) a pitch motion of PITCH degrees.

QUASI.ext (1,2) Vector defining the magnitude of the roll and pitch motions (in degrees) for which the maximum engine displacements are calculated.

Column 1: ROLL
Column 2: PITCH

Output files:

VVESACTIVE.15 Quasi-static displacements

Output format:

Line 1:
Column 1: Maximum displacements caused by ship roll
Column 2: Tx for centre of mass caused by ship roll
Column 3: Ty for centre of mass caused by ship roll
Column 4: Tz for centre of mass caused by ship roll
Column 5: Rx for centre of mass caused by ship roll (degrees)

Column 6: Ry for centre of mass caused by ship roll (degrees)
Column 7: Rz for centre of mass caused by ship roll (degrees)
Line 2:
Column 1: Maximum displacements caused by ship pitch
Column 2: Tx for centre of mass caused by ship pitch
Column 3: Ty for centre of mass caused by ship pitch
Column 4: Tz for centre of mass caused by ship pitch
Column 5: Rx for centre of mass caused by ship pitch (degrees)
Column 6: Ry for centre of mass caused by ship pitch (degrees)
Column 7: Rz for centre of mass caused by ship pitch (degrees)

2.7 Active Control

The work on the development of the active control algorithm was conducted by Alain Berry, Professor; Patrice Masson, Assistant Professor; Stéphane Renault, Research Associate from the Department of Mechanical Engineering at the University of Sherbrooke.

The active control work extends the existing VVES code which simulates the natural frequencies and transmissibility of a multi-component vibration isolation system that holds a massive rigid body over a flexible foundation. The work proposed here will extend the capabilities of the VVES software by simulating the action of active control forces on the system to increase the isolation provided by a purely passive system. The software allows the inclusion of multiple control forces on the physical system, simulates the measurement of error sensors at various locations on the physical system, and simulates the action of a real-time control system to minimize the error sensor signals using the control forces. In this project, the GAUS acts as a sub-contractor of the prime contractor, VIBE-TEK RESEARCH LTD which produced the VVES software.

Control forces can be applied in all six degrees of freedom (three orthogonal forces and three orthogonal torques) between any two points on the engine or the raft. Two distinct feedforward control configurations are implemented in two distinct program sets. The first one allows vibration control simulation where the reference signals are not affected by the signal coming from the actuators (see Figure 1). This would be the case with a reference signal obtained from a tachometer mounted on the engine rotor for example. The second configuration allows vibration control simulation where the reference signals are measured by sensors which can be affected by the signal coming from the actuators (see Figure 2). This would be the case with a reference signal measured using an accelerometer mounted on the engine for example.

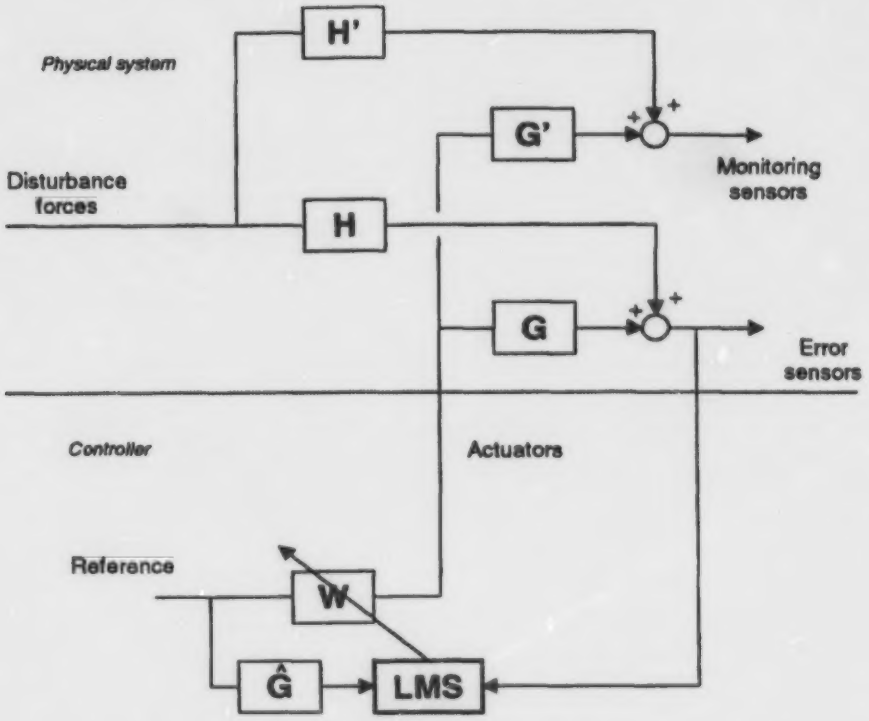


Figure 1: Feedforward control approach with direct access to the reference signals.

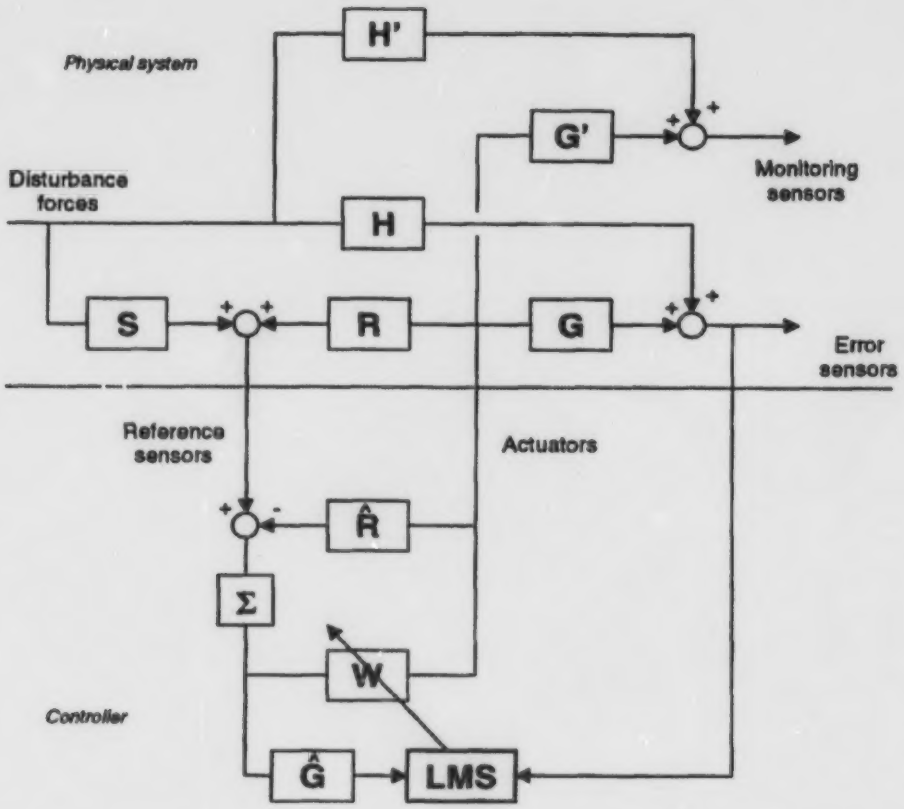


Figure 2: Feedforward control approach using sensors to measure the reference signals.

The displacement along all six degrees of freedom (Tx, Ty, Tz, Rx, Ry, Rz), as could be measured by appropriate sensors, are provided as error, reference or monitoring signals. The control actuators are force actuators.

The programs require Matlab 5.3, the Signal Processing Toolbox and the DSP Blockset which are available from Mathworks Inc.

2.7.1 Details of the Programs

Two sets of programs are provided to perform the control simulation of the real-time controller using the VVES program.

1. The program **AVC1.mdl** is the Simulink program for active vibration control simulation with a feedforward controller where the reference signals are not affected by the signal coming from the actuators. The control approach implemented in this program is presented in Figure 1. The Simulink block code is presented in the Appendix.

   The Matlab program **INIAVC1.m** is used to compute the matrices **H, H', G, G'** necessary for the simulation with AVC1.mdl. These matrices represent impulse

response functions (the time-domain inverse Fourier transform of transfer functions), and their size is determined by the number of disturbance forces, error sensors, monitoring sensors and actuators. More specifically:

H' is a matrix containing the impulse response functions between monitoring sensors and disturbance forces,

G is a matrix containing the impulse response functions between error sensors and forces generated by actuators,

G' is a matrix containing the impulse response functions between monitoring sensors and forces generated by actuators,

$\hat{\mathbf{G}}$ is the estimate of **G**. For the simulation $\hat{\mathbf{G}} = \mathbf{G}$.

W is a matrix containing the finite impulse response adaptive control filters.

The program computes the impulse response functions by calling first the program VVES_AVC.m (with ncase = 1) and the program FRFSISO.m supplied in the active control package.

Input data files for VVES_AVC (details provided in the VVES user guide):

NODE.ext	ELEMENT.ext	CONSTR.ext
BEAMS.ext	MASSES.ext	ENGINE.ext
RAFT.ext	VISCO.ext	NXYZ.ext
COORD.ext	RESP.ext	FORCE.ext
FTRANS.ext	DTRANS.ext	QUASI.ext

Input data files for INIAVC1.m:

PARAM.ext	DISTURB.ext	ACTUATOR.ext
ERRSENS.ext	MONSENS.ext	

The Matlab program **POSTAVC1.m** is used to process the results obtained from the simulation and write the results in files.

Output files:

ERR_WOC.txt	ERR_WC.txt	MON_WOC.txt
MON_WC.txt	DISTU.txt	CMD.txt

2. The program **AVC2.mdl** is the Simulink program for active vibration control simulation with a feedforward controller where the reference signals are measured by sensors which can be affected by the signal coming from the actuators. The control approach implemented in this program is presented in Figure 2. The Simulink block code is presented in the Appendix.

The Matlab program **INIAVC2.m** is used to compute the matrices **H, H', G, G', S** and **R** necessary for the simulation with AVC2.mdl. These matrices represent impulse response functions (the time-domain inverse Fourier transform of transfer functions), and their size is determined by the number of disturbance forces, reference sensors, error sensors, monitoring sensors and actuators. More specifically:

H is a matrix containing the impulse response functions between error sensors and disturbance forces,
H' is a matrix containing the impulse response functions between monitoring sensors and disturbance forces,
G is a matrix containing the impulse response functions between error sensors and forces generated by actuators,
G' is a matrix containing the impulse response functions between monitoring sensors and forces generated by actuators,
S is a matrix containing the impulse response functions between reference sensors and disturbance forces,
R is a matrix containing the impulse response functions between reference sensors and forces generated by actuators,
$\hat{\mathbf{G}}$ is the estimate of **G**. For the simulation $\hat{\mathbf{G}} = \mathbf{G}$.
$\hat{\mathbf{R}}$ is the estimate of **R**. For the simulation $\hat{\mathbf{R}} = \mathbf{R}$.
W is a matrix containing the finite impulse response adaptive control filters.

The program computes the impulse response functions by calling first the program VVES_AVC.m (with ncase = 1) and the program FRFSISO.m supplied in the active control package.

Input data files for VVES_AVC (details provided in the VVES user guide):

NODE.ext	ELEMENT.ext	CONSTR.ext
BEAMS.ext	MASSES.ext	ENGINE.ext
RAFT.ext	VISCO.ext	NXYZ.ext
COORD.ext	RESP.ext	FORCE.ext
FTRANS.ext	DTRANS.ext	QUASI.ext

Input data files for INIAVC2.m:

PARAM.ext	DISTURB.ext	ACTUATOR.ext
ERRSENS.ext	MONSENS.ext	REFSENS.ext

The Matlab program **POSTAVC2.m** is used to process the results obtained from the simulation and write the results in files.

Output files:

ERR_WOC.txt	ERR_WC.txt	MON_WOC.txt
MON_WC.txt	DISTU.txt	CMD.txt

2.7.2 Details of output files

ERR_WOC.txt

Column 1 : Time (seconds)
Column 2 to (Ns+1) : Error signals without control
Column Ns+2 : Sum of the of the squared error signals, without control
(Ns is the number of error sensors)

ERR_WC.txt

Column 1 : Time (seconds)
Column 2 to (Ns+1) : Error signals with control
Column Ns+2 : Sum of the squared error signals, with control

MON_WOC.txt

Column 1 : Time (seconds)
Column 2 to (Nm+1) : Monitoring signals without control
Column Nm+2 : Sum of the of the squared monitoring signals, without control
(Nm is the number of monitoring sensors)

MON_WC.txt

Column 1 : Time (seconds)
Column 2 to (Nm+1) : Monitoring signals with control
Column Nm+2 : Sum of the of the squared monitoring signals, with control

DISTU.txt

Column 1 : Time (seconds)
Column 2 to (Nd+1) : Disturbance signals
(Nd is the number of disturbing forces)

CMD.txt

Column 1 : Time (seconds)
Column 2 to (Nd+1) : Control signals

3. Details of input files

The programs uses information stored in files with the suffix .ext to define the required system parameters. Different programs require different input data. The requirements of the simulation programs are given in section 2.7.1.

3.1 Modeling of the engine

The engine is modeled as a rigid body in terms of the location of the centre of gravity; its mass; and its principal moments of inertia about the centre of mass. (It is assumed that the principal directions coincide with the global coordinate axes)

ENGINE.ext (1:1, 1:8+NSP) vector defining engine parameters, where NSP is the number of engine support points:

Column 1: M (kg)
Column 2: I_{xx} mass moment of inertia about X axis (kg-m^2)
Column 3: I_{yy} mass moment of inertia about Y axis (kg-m^2)
Column 4: I_{zz} mass moment of inertia about Z axis (kg-m^2)
Column 5: length of engine in the X-direction (only used for graphics)
Column 6: length of engine in the Y-direction (only used for graphics)
Column 7: length of engine in the Z-direction (only used for graphics)
Column 8: nodal number of center of mass of the engine
Column 9: engine support node number 1
Column 10: engine support node number 2
....
Column 8+NSP: engine support node number NSP

3.2 Modeling of the foundation raft

Two options are provided for the modeling of the foundation raft. It can be modeled as a rigid body with its inertial characteristics provided by the user and its geometric characteristics defined in terms of nodal points in NODE.ext; or it can be modeled in terms of elastic finite element beam elements in order to asses the effects of its flexibility.

3.2.1 Modeling of the raft as a rigid body

The raft is modeled as a rigid body in terms of the location of its centre of gravity; its mass; and its principal moments of inertia about the centre of mass. (It is assumed that the principal directions coincide with the global coordinate axes)

RAFT.ext (1:1, 1:7+NR) vector defining rigid raft parameters, where NR is the number of raft support points. NR=NRT+NRB where NRT= number of isolators connected to the top of the raft and NRB= number connected to the bottom of the raft. :

Column 1: M (kg). (If the flexible raft option is requiredd, set this number to zero)
Column 2: I_{xx} mass moment of inertia about X axis (kg-m^2)
Column 3: I_{yy} mass moment of inertia about Y axis (kg-m^2)
Column 4: I_{zz} mass moment of inertia about Z axis (kg-m^2)
Column 5: length of raft in the X-direction (only used for graphics)
Column 6: length of raft in the Y-direction (only used for graphics)
Column 7: node number of center of mass of the raft
Column 8: node number of connection 1 between top of the raft and engine
Column 9: node number of connection 2 between top of the raft and the engine on the raft top
....
Column 7+NRT: node number of connection NRT between top of the raft and the engine
Column 7+NRT+1: minus node number of connection 1 between underside of the raft and the ground
Column 7+NRT+2: minus node number of connection 2 between underside of the raft and the ground
⋮
Column 7+NRT+NRB: minus node number of connection NRB between underside of the raft and the ground

3.2.2 Modeling of the raft as a flexible body

The raft can also be modeled as a flexible frame using elastic beam elements. In this modeling it is important to include the effect of the depth of the beam members when specifying the properties of the isolators in COORD.ext. This is done by appropriate

definition of the local coordinates specifying the top and bottom of the condensed one dimensional representation of the isolator.

In order to invoke this option the value of the mass in RAFT.ext should be set to zero, otherwise the rigid raft option will be chosen.

3.3 Modeling of the isolators

The isolators are modeled differently in different programs.

For finite element analysis conducted internally to the program three isolator shapes are permissible: (i): a rectilinear solid; (ii): a solid cone; and (iii): a hollow cone. (An external finite element analysis can be conducted and the data imported to the program through Kve.ext and Mve.ext)

The parameters defining the shape of the isolators and the two local nodal coordinates that define the idealized one-dimensional condensed representation of the isolator are stored in COORD.ext. The origin of the local coordinates used for defining the isolators and their condensed representation is the centre of the base of the isolator and the local z direction of the isolator is that normal to the surface of the isolator.

It is assumed that the isolators can have different geometric and viscoelastic properties and the total number of different isolators is N. An isolator is defined to be different if it has a different geometry by virtue of its shape or by virtue of having different local coordinates specifying its end points, or by virtue of having different constitutive properties. VISCO.ext contains N rows that define the corresponding viscoelastic parameters for each of these isolators.

COORD.ext (1:5,1:3×N) matrix defining isolator geometry characteristics.

For each three columns:
Row 1: Defines geometric shape of the isolator
coord(1,1)=1, for rectilinear solid.
{side length in x-direction: a1(base); a2(top).
side length in y-direction: b1(base); b2(top).
height in z-direction: L}

coord(1,1)=2, for hollow cylinder.
{inner diameter: d1(base); d2(top);
outer diameter: D1(base); D2(top);
height in z-direction: L}

coord(1,1)=3, for solid cylinder.
{diameter: D1(base); D2(top);
height in z-direction: L}

Row 2: refers to the bottom of isolator:
Column 1: a1: base for rectangular solid
d1: base for hollow cylinder
0 for solid cylinder
Column 2: b1: base for rectangular solid
D1: base for both hollow cylinder and solid cylinder
Column 3: local z-coordinate of base

Row 3: refers to the top of isolator:
Column 1: a2: top for rectangular solid
d2 : top for hollow cylinder
0 : for solid cylinder
Column 2: b2: top for rectangular solid;
D2: top for both hollow cylinder and solid cylinder;
Column 3: local z-coordinate of top

Row 4: local coordinates of base of condensed isolator. This point will be rigidly connected to the top of the rigid raft or ground, or to the node of a beam element in the case of a flexible raft.

Row 5: local coordinates of top of condensed isolator. This point will be rigidly connected to the engine, or to the bottom of the rigid raft or to the node of a beam element in the case of a flexible raft.

{Rows 4 and 5 refer to the nodal locations of the one dimensional condensed representation of the isolator and refer to an origin at the centre of the base of the isolator.}

The finite element mesh discretization of the isolator is defined in NXYZ.ext.

NXYZ.ext (1:N, 1:3) matrix defining isolator finite element idealizations for the N different isolators defined in COORD.ext.

For rectilinear solid:
Column 1: number of elements in the x-direction
Column 2: number of elements in the y-direction
Column 3: number of elements in the z-direction

For solid and hollow cone:
Column 1: number of elements in the radial direction
Column 2: number of elements in the circumferential direction
Column 3: number of elements in the z-direction

The viscoelastic properties of the isolator are defined in VISCO.ext

VISCO.ext (1:N, 1:8) matrix defining the isolator viscoelastic properties. The total number is N and each row corresponds to an isolator property defined in COORD.ext.

i) for viscoelastic characteristics of the isolators defined in terms of the rational polynomial (RP) formula. For complex Poisson's ratio case, there are two complex moduli $G_1^*(i\omega) = G_1(1 - \sum_{n=1}^{2} \frac{a_{1n}}{i\omega + b_n})$

$G_2^*(i\omega) = G_2(1 - \sum_{n=1}^{2} \frac{a_{2n}}{i\omega + b_n})$

The relationships among $G_1^*(i\omega)$, $G_2^*(i\omega)$ and complex Young's modulus $E^*(i\omega)$, complex Poisson's Ratio $\nu^*(i\omega)$ are

$$G_1^*(i\omega) = \frac{E^*(i\omega)}{2[1+\nu^*(i\omega)]} \qquad \text{and} \qquad G_1^*(i\omega) = \frac{E^*(i\omega)}{3[1-2\nu^*(i\omega)]}$$

Column 1: 1 (for RP model)
Column 2: G_1 (N/m^2)
Column 3: a_{11} (Hz)
Column 4: b_1 (Hz)
Column 5: a_{12} (Hz)
Column 6: b_2 (Hz)
Column 7: mass density of isolator (kg/m^3)
Column 8: Poisson's Ratio at zero frequency point
Column 9: G_2 (N/m^2)
Column 10: a_{21} (Hz)
Column 11: a_{22} (Hz)

ii) for viscoelastic characteristics of isolators defined in terms of the Havrilak Negami (HN) model:

$$G_1^*(i\omega) = G_{1\infty} + (G_{10} - G_{1\infty})[1 + (i\omega\tau_1)^{\alpha_1}]^{-\beta_1}$$
$$G_2^*(i\omega) = G_{2\infty} + (G_{20} - G_{2\infty})[1 + (i\omega\tau_2)^{\alpha_2}]^{-\beta_2}$$

Column 1: 2 (for HN model)
Column 2: α_1 (dimensionless)
Column 3: β_1 (dimensionless)
Column 4: τ_1 (1/Hz)
Column 5: G_{10} (N/m^2)
Column 6: $G_{1\infty}$ (N/m^2)
Column 7: mass density (kg/m^3)
Column 8: Poisson's Ratio at zero frequency point
Column 9: α_2 (dimensionless)
Column 10: β_2 (dimensionless)
Column 11: τ_2 (1/Hz))
Column 12: G_{20} (N/m^2)
Column 13: $G_{2\infty}$ (N/m^2)

iii) for viscoelastic characteristics of isolators defined in terms of experimental data provided in DVISCO.ext:

Column 1: 3 (for experimental data)
Column 2: 0
Column 3: 0
Column 4: 0
Column 5: 0
Column 6: 0
Column 7: mass density (kg/m^3)
Column 8: Poisson's Ratio at zero frequency

The required viscoelastic properties are retrieved from this file by specifying the required row number in ELEMENT.ext.

DVISCO.ext (1:Nfreq, 4×N+1) matrix defining the complex modulus data that is to be used to model the behaviour of the isolators. Nfreq is the number of data points.

Column 1: Frequency in Hz
Column 2: Real part of complex modulus $G_1^*(i\omega)$ data of the first isolator
Column 3: Imaginary part of complex modulus $G_1^*(i\omega)$ data of the first isolator
Column 4: Real part of complex modulus $G_2^*(i\omega)$ data of the first isolator
Column 5: Imaginary part of complex modulus $G_2^*(i\omega)$ data of the first isolator
...
Column 4×N-2: Real part of complex modulus $G_1^*(i\omega)$ data of the last isolator
Column 4×N-1: Imaginary part of complex modulus $G_1^*(i\omega)$ data of the last isolator
Column 4×N: Real part of complex modulus $G_2^*(i\omega)$ data of the last isolator
Column 4×N+1: Imaginary part of complex modulus $G_2^*(i\omega)$ data of the last isolator

VISCODS.ext (1:N, 1:8) matrix defining the isolator viscoelastic properties. The total number is N and each row corresponds to an isolator property defined in COORD.ext.

Column 1: 1
Column 2: 0
Column 3: 0
Column 4: 0
Column 5: 0
Column 6: 0
Column 7: mass density of isolator (kg/m^3)
Column 8: Poisson's Ratio at zero frequency point

KVE.ext (1:N, 1:6) vector containing the parameters that define the diagonal elements of the condensed stiffness matrix of the isolators. If this data is known it can be imported into the program.
For each row:
Column 1: K11(N/m)
Column 2: K22(N/m)
Column 3: K33(N/m)
Column 4: K44(N-m)
Column 5: K55(N-m)
Column 6: K66(N-m)

Knn is the stiffness of the isolator in the n th direction due to displacement or rotation in the n th direction. n=1,2,3 represents displacements in the x, y, and z directions and n=4,5,6 represents rotations about the x y and z axes.

MVE.ext (1:N, 1:6) vector containing the parameters that define the diagonal elements of the condensed mass matrix of the isolator. If this data is known it can be imported into the program.

For each row:
Column 1: M11 (Kg)
Column 2: M22 (Kg)
Column 3: M33 (Kg)
Column 4: M44 (Kg-m^2)
Column 5: M55 (Kg-m^2)
Column 6: M66 (Kg-m^2)

Mnn is the mass (rotational inertia) of the isolator in the n th direction. n=1,2,3 represents displacements in the x, y, and z directions and n=4,5,6 represents rotations about the x, y and z axes.

File OFHZ.ext (MNUM,1) is a data file of frequency points defining the frequencies (Hz) at which the 4-pole parameter data has been measured for VVES4POLE.m . MNUM is the total number of frequency points.

File FPX1.ext (1:MNUM,1:8) are the measured four-pole parameter matrices measured along the x-direction for isolator layer 1.
Each row corresponds to the frequency defined in the same row of OFHZ.ext

Column 1: real part of this 4-pole parameter matrix element (1,1)
Column 2: imaginary part of this 4-pole parameter matrix element (1,1)
Column 3: real part of this 4-pole parameter matrix element (1,2)
Column 4: imaginary part of this 4-pole parameter matrix element (1,2)
Column 5: real part of this 4-pole parameter matrix element (2,1)
Column 6: imaginary part of this 4-pole parameter matrix element (2,1)
Column 7: real part of this 4-pole parameter matrix element (2,2)
Column 8: imaginary part of this 4-pole parameter matrix element (2,2)

File FPY1.ext (1:MNUM,1:8) are the four-pole parameter matrices measured along the y-direction for isolator layer 1.

Each row corresponds to the frequency defined in the same row of OFHZ.ext

Column 1: real part of this 4-pole parameter matrix element (1,1)
Column 2: imaginary part of this 4-pole parameter matrix element (1,1)
Column 3: real part of this 4-pole parameter matrix element (1,2)
Column 4: imaginary part of this 4-pole parameter matrix element (1,2)
Column 5: real part of this 4-pole parameter matrix element (2,1)
Column 6: imaginary part of this 4-pole parameter matrix element (2,1)
Column 7: real part of this 4-pole parameter matrix element (2,2)
Column 8: imaginary part of this 4-pole parameter matrix element (2,2)

File FPZ1.ext (1:MNUM,1:8) are the four-pole parameter matrices measured along z-direction for isolator layer 1.

Column 1: real part of this 4-pole parameter matrix element (1,1)
Column 2: imaginary part of this 4-pole parameter matrix element (1,1)
Column 3: real part of this 4-pole parameter matrix element (1,2)
Column 4: imaginary part of this 4-pole parameter matrix element (1,2)
Column 5: real part of this 4-pole parameter matrix element (2,1)
Column 6: imaginary part of this 4-pole parameter matrix element (2,1)
Column 7: real part of this 4-pole parameter matrix element (2,2)
Column 8: imaginary part of this 4-pole parameter matrix element (2,2)

File FPX2.ext (1:MNUM,1:8) are the four-pole parameter matrices measured along the x-direction for isolator layer 2.

Column 1: real part of this 4-pole parameter matrix element (1,1)
Column 2: imaginary part of this 4-pole parameter matrix element (1,1)
Column 3: real part of this 4-pole parameter matrix element (1,2)
Column 4: imaginary part of this 4-pole parameter matrix element (1,2)
Column 5: real part of this 4-pole parameter matrix element (2,1)
Column 6: imaginary part of this 4-pole parameter matrix element (2,1)
Column 7: real part of this 4-pole parameter matrix element (2,2)
Column 8: imaginary part of this 4-pole parameter matrix element (2,2)

File FPY2.ext (1:MNUM,1:8) are the four-pole parameter matrices measured along the y-direction for isolator layer 2.

Column 1: real part of this 4-pole parameter matrix element (1,1)
Column 2: imaginary part of this 4-pole parameter matrix element (1,1)
Column 3: real part of this 4-pole parameter matrix element (1,2)
Column 4: imaginary part of this 4-pole parameter matrix element (1,2)
Column 5: real part of this 4-pole parameter matrix element (2,1)
Column 6: imaginary part of this 4-pole parameter matrix element (2,1)
Column 7: real part of this 4-pole parameter matrix element (2,2)
Column 8: imaginary part of this 4-pole parameter matrix element (2,2)

File FPZ2.ext (1:MNUM,1:8) are the four-pole parameter matrices measured along z-direction for isolator layer 2.

Column 1: real part of this 4-pole parameter matrix element (1,1)
Column 2: imaginary part of this 4-pole parameter matrix element (1,1)
Column 3: real part of this 4-pole parameter matrix element (1,2)
Column 4: imaginary part of this 4-pole parameter matrix element (1,2)
Cclumn 5: real part of this 4-pole parameter matrix element (2,1)
Column 6: imaginary part of this 4-pole parameter matrix element (2,1)
Column 7: real part of this 4-pole parameter matrix element (2,2)
Column 8: imaginary part of this 4-pole parameter matrix element (2,2)

NFREQ.ext (1, 1:3) matrix defining the frequency range and number of frequency points
used in the computation of the response output from VVES4POLE.m

Column 1: the lowest frequency (Hz) for calculation
Column 2: the highest frequency (Hz) for calculation
Column 3: the number of discrete frequency points used in the analysis

The program assumes that the experimental data obtained in the x, y and z directions correspond to cases where the isolator is free to displace in the direction of the applied force but the ends of the isolator are restrained from rotation.

File DIRECTSTIFF.ext (1:MNUM,1:13) Contains the measured dynamic stiffness of each layer of isolators (each layer is assumed to have only one type of isolator) measured in the x, y and z directions.

Column 1: frequency in Hz
Column 2: real part of complex stiffness k11 for isolator layer 1
Column 3: imaginary part of complex stiffness k11 for isolator layer 1
Column 4: real part of complex stiffness k22 for isolator layer 1
Column 5: imaginary part of complex stiffness k22 for isolator layer 1
Column 6: real part of complex stiffness k33 for isolator layer 1
Column 7: imaginary part of complex stiffness k33 for isolator layer 1
Column 8: real part of complex stiffness k11 for isolator layer 2
Column 9: imaginary part of complex stiffness k11 for isolator layer 2
Column 10: real part of complex stiffness k22 for isolator layer 2
Column 11: imaginary part of complex stiffness k22 for isolator layer 2
Column 12: real part of complex stiffness k22 for isolator layer 2
Column 13: imaginary part of complex stiffness k22 for isolator layer 2

The program assumes that the experimental data obtained in the x, y and z directions correspond to cases where the isolator is free to displace in the direction of the applied force but the ends of the isolator are restrained from rotation.

3.4 Modeling the geometry of the complete system

The nodal coordinates that define the supporting beam elements, the centre of mass of the engine, the centre of mass of the raft (if required) and the end points of the isolators are contained in the file NODE.ext .

NODE.ext (1:NNODE+1, 1:4) matrix defining nodal coordinates

The first row:
Column 1: NNODE; Column 2 to Column 4: 0 0 0
The other rows:
Column 1: x coordinate
Column 2: y coordinate
Column 3: z coordinate
Column 4: node number

When running VVES4POLE.m an extra node needs to be defined to specify the base point about which the base velocities are input.

CONSTR.ext (1:NNODE, 1:7) matrix defining the nodal constraint characteristics for all the nodes.
Note that this file is not required when running VVES4POLE.m

Nodes that are attached directly to the engine are considered to be fixed by virtue of the fact that their displacement is entirely determined by the displacement of the engine. Thus if node 7 is an engine support node the appropriate line would be [1 1 1 1 1 1 7].

Nodes that are attached directly to the raft are considered to be fixed by virtue of the fact that their displacement is entirely determined by the displacement of the raft. Thus if node 9 is attached to the raft the appropriate line would be [1 1 1 1 1 1 9].

Nodes that are attached to "ground" require a special designation and are identified by the negative sign. Thus if node 20 were a simply supported node attached to ground it would be designated by [-1 -1 -1 0 0 0 20].

Other nodes are free, so if 5 were such a node its designation would be:
[0 0 0 0 0 0 5].

Column 1:	x direction translation	=0 free;	=1 fixed ;	= -1 as a supported DOF;
Column 2:	y direction translation	=0 free;	=1 fixed;	= -1 as a supported DOF;
Column 3:	z direction translation	=0 free;	=1 fixed;	= -1 as a supported DOF;
Column 4:	x direction rotation	=0 free;	=1 fixed;	= -1 as a supported DOF;
Column 5:	y direction rotation	=0 free;	=1 fixed;	= -1 as a supported DOF;
Column 6:	z direction rotation	=0 free;	=1 fixed;	= -1 as a supported DOF;
Column 7:	node number			

The definition of the nodal coordinates for each of the elements of the system is given in ELEMENT.ext

ELEMENT.ext (1:NELEM+1, 1:8) matrix defining the element characteristics

The first row:
Column 1: NELEM; Column 2 to Column 6: 0 0 0 0 0
The other rows:
Column 1: type of element
= 1 - for viscoelastic isolator: element matrices computed internally.
=2 - for viscoelastic isolator: element matrices supplied by user
=3 - beam element
=4 - mass element
Column 2: reference number defining the row in which the required element properties are found:
1 to N - for isolator elements from VISCO.ext [different row numbers define both different materials and different material representations]
1 to NBEAM - for beam elements from BEAMS.ext
1 to NMASS - for mass elements from MASSES.ext

Column 3: node number 1 of this element/isolator
Column 4: node number 2 of this element/isolator
Column 5: node number 3 defining the orientation of this element/isolator, which can be a real node of the model or a dummy node. This node together with node numbers 1 and 2 define a plane which is the local y-z plane of the element.
Column 6: global element number

ELEMENT4.ext (1:NELEM+1, 1:8) matrix defining the element characteristics when using the program VVES4POLE.m

The first row:
Column 1: NELEM; Column 2 to Column 6: 0 0 0 0 0
The other rows:
Column 1: type of element
= 1 - for viscoelastic isolator: element matrices computed internally using DMESH.M
=2 - for viscoelastic isolator: element matrices supplied by user
=3 - beam element
=4 - mass element
Column 2: reference number defining the row in which the required element properties are found:
1 to N - for isolator elements from VISCO.ext [different row numbers define both different materials and different material representations]
1 to NBEAM - for beam elements from BEAMS.ext
1 to NMASS - for mass elements from MASSES.ext

Column 3: node number 1 of this element/isolator
Column 4: node number 2 of this element/isolator
Column 5: node number 3 defining the orientation of this element/isolator, which can be a real node of the model or a dummy node. This node together with node numbers 1 and 2 define a plane which is the local y-z plane of the element.
Column 6: node number 4 defining the node rigidly connected with node number 1
Column 7: node number 5 defining the node rigidly connected with node number 2
Column 8: global element number

In columns 7 and 8 the nodes rigidly connected to the ends of the isolators will correspond to the center of mass of the engine or the raft, or the reference point defining the base.

The definition of the element properties of the elastic beams are defined in BEAMS.ext.

BEAMS.ext (1:NBEAM, 1:7) matrix defining material and geometric characteristics of the elastic beams. The properties are specified with respect to local axes (x,y,z) where z is the longitudinal axis and y is the vertical axis. NBEAM are the number of different beam types specified.

Column 1: Young's modulus (N/m^2)
Column 2: Poisson's ratio
Column 3: Mass density (kg/m^3)
Column 4: Transverse area of the beam (m^2)
Column 5: Second moment of area around the local y-axis(I_{yy}) (m^4)
Column 6: Second moment of area around the local x-axis (I_{xx}) (m^4)
Column 7: Torsional constant of area around the local z-axis (J) (m^4)

The definition of any mass elements (other than the engine and the rigid raft) is given in MASSES.ext. These mass elements are assumed to be concentrated at a point. The properties are defined with respect to the same local axes as for the beams.

MASSES.ext (1:NMASS, 1:7)
Column 1: M (kg)
Column 2: I_{xx} (kg-m^2)
Column 3: I_{yy} (kg-m^2)
Column 4: I_{zz} (kg-m^2)

3.5 Definition of the type of forced response analysis to be conducted

Each of the programs requires different input data in order to determine specified forced response characteristics of the system. Table 3 summarizes the data files required for each program.

Response	Input File	vvesanalysis	vvesdynstff	vvesactive
Frequency Response		Frfunanalysis.m	Frfundynstff.m	Frfunactive.m
	Force.ext	X	X	X
	Resp.ext	X	X	X
Force Transmissibility		Ftrssanalysis.m	Ftrssdynstff.m	Ftrssactive.m
	Ftrans.ext	X	X	X
	Tftrans.ext	X	X	X
Displacement Transmissibility		Dtrssanalysis.m	Dtrssdynstff.m	Dtrssactive.m
	Dtrans.ext	X	X	X

Table 3 Required Data Files for Response Analysis

3.5.1 Frequency Response

FORCE.ext (1, 1:10) matrix defining the node and degree of freedom that is loaded:

Column 1: the node number loaded or "0" if another load point on engine is required
Column 2: x coordinate of the applied force, or "0" if applied at any node
Column 3: y coordinate of the applied force, or "0" if applied at any node
Column 4: z coordinate of the applied force, or "0" if applied at any node
Column 5: force along the x-direction =1 if loaded, = 0 otherwise
Column 6: force along the y-direction =1 if loaded, = 0 otherwise
Column 7: force along the z-direction =1 if loaded, = 0 otherwise
Column 8: moment around the x-direction =1 if loaded, = 0 otherwise
Column 9: moment around the y-direction =1 if loaded, = 0 otherwise
Column10: moment around the z-direction =1 if loaded, = 0 otherwise

The frequency range over which the response is required is given in RESP.ext.
RESP.ext (1: 1, 1:4) vector defining frequency range and the node at which the frequency response characteristic is required and the specification of the nodes for which the velocity ratio is required.

Column 1: the lowest frequency (Hz) for frequency response calculation
Column 2: the highest frequency (Hz) for frequency response calculation
Column 3: the number of discrete frequency points
Column 4: node number for which the frequency response calculation is required
Column 5: node number n1
Column 6: node number n2
Program calculates (velocity of n1/velocity of n2 for all six nodal degrees of freedom). If n1=0 no calculation is performed.

3.5.2. Force Transmissibility

Two options are available: one computes the force transmissibility between an applied force and a specific isolator and the other computes the transmissibility between a specified force and the total force transmitted through all the isolators in a specified direction.

3.5.2.1 Force Transmissibility

This program calculates force transmissibility functions between loads applied to the engine and the corresponding forces transmitted to a specified foundation node.
The definition of the required parameters is contained in FTRANS.ext

FTRANS.ext (1:1, 1:8) vector containing the parameters that define the force transmissibility calculations. The force applied to the engine can be applied at any point not just at the node corresponding to the centre of mass.

Column 1: support node number at which the transmissibility is calculated
Column 2: the node number of the centre of mass or "0" if another load point is required
Column 3: x coordinate of the applied force, or "0" if applied at centre of mass
Column 4: y coordinate of the applied force, or "0" if applied at centre of mass
Column 5: z coordinate of the applied force, or "0" if applied at centre of mass
Column 6: the lowest frequency (Hz) for frequency response calculation
Column 7: the highest frequency (Hz) for frequency response calculation
Column 8: the number of discrete frequency points

3.5.2.2 Total Force Transmissibility

The program calculates the total force transmissibility functions between loads applied to the engine and the corresponding forces transmitted to all foundation nodes.
The definition of the required parameters is contained in TFTRANS.ext

TFTRANS.ext (1:1, 1:7) vector containing the parameters that define the force transmissibility calculations. The force applied to the engine can be applied at any point not just at the node corresponding to the centre of mass.

Column 1: the node number of the centre of mass or "0" if another load point is required
Column 2: x coordinate of the applied force, or "0" if applied at centre of mass
Column 3: y coordinate of the applied force, or "0" if applied at centre of mass
Column 4: z coordinate of the applied force, or "0" if applied at centre of mass
Column 5: the lowest frequency (Hz) for frequency response calculation
Column 6: the highest frequency (Hz) for frequency response calculation
Column 7: the number of discrete frequency points

3.5.3 Displacement Transmissibility

The program calculates displacement transmissibility functions between rigid body displacements and rotations applied to the engine support points, with reference to a user specified point, and the corresponding displacements transmitted to the engine. Using this program enables the effect of ship motion, defined about some arbitrary point, on the response of the engine to be calculated.
The definition of the required parameters is contained in DTRANS.ext

DTRANS.ext (1:1, 1:7) vector containing the parameters that define the displacement transmissibility calculations.

Column 1: number of nodes rigidly connected to the base structure
Column 2: base node number 1
⋮
Column BNOD+1: base node number BNOD
Column BNOD+2: node number at which displacement transmissibility functions are to be calculated
Column BNOD+3: x coordinate of the origin of the support motion
Column BNOD+4: y coordinate of the origin of the support motion
Column BNOD+5: z coordinate of the origin of the support motion
Column BNOD+6: the lowest frequency (Hz) for calculation
Column BNOD+7: the highest frequency (Hz) for calculation
Column BNOD+8: the number of discrete frequency points

3.5.4 Active Control

PARAM.ext

Specifies the parameters to be used in the control algorithm

Column 1 : Sampling frequency (Fe),
Column 2 : Number of coefficients for the Finite Impulse Response filters representing the various impulse response functions between the excitations and the sensors (Nc),

Column 3 : Time period of the Simulink simulation (Time_len),
Column 4 : Convergence coefficient of the LMS (Lambda),
Column 5 : Number of coefficients for the control filters (Nco).

DISTURB.ext

Specifies the disturbance (primary forces) applied to the system.

Line 1 :
Column 1 : Number of disturbance forces (Nd),
Column 2 : Fundamental frequency (F0),
Column 3 : Number of harmonics (including the fundamental frequency) (Nh),
Other Columns : 0
Line 2 to Nd+1 :
Column 1 : Node number of the center of mass of the engine if the disturbance is applied on the engine, or node number of the center of mass of the raft if the disturbance is applied on the raft.
Column 2: x coordinate of the applied force
Column 3: y coordinate of the applied force
Column 4: z coordinate of the applied force
Column 5: direction of the applied force
1 - force along the x-direction
2 - force along the y-direction
3 - force along the z-direction
4 - moment around the x-direction
5 - moment around the y-direction
6 - moment around the z-direction
Columns 6 to 5+2.Nh :
Even columns : Magnitude of the force for each harmonic,
Odd columns: Phase of the force for each harmonic.

ACTUATOR.ext

Specifies the control actuator configuration.

Line 1 :
Column 1 : Number of control forces (Na),
Column 2 : Type of actuators (TA) :
0 – conventional actuators,
1 – actuators between the raft and the engine,
Other columns : 0,

If TA = 0
Line 2 to Na+1 :
Column 1 : Node number of the center of mass of the engine if the actuator is placed on the engine, or node number of the center of mass of the

raft if the actuator is placed on the raft.
Column 2: x coordinate of the applied force
Column 3: y coordinate of the applied force
Column 4: z coordinate of the applied force
Column 5: direction of the applied force
1 - force along the x-direction
2 - force along the y-direction
3 - force along the z-direction
4 - moment around the x-direction
5 - moment around the y-direction
6 - moment around the z-direction

else If TA = 1
Line 2 to Na+1 :
Column 1: x coordinate of the actuator,
Column 2: y coordinate of the actuator.

REFSENS.ext

Specifies the reference sensor configuration (applies only to AVC2.mdl).

Line 1 :
Column 1 : Number of reference sensors (Nr),
Columns 2 to 5 : 0,
Line 2 to (Nr+1) :
Column 1 : Node number of the center of mass of the engine if the reference sensor is placed on the engine, or node number of the center of mass of the raft if the reference sensor is placed on the raft.
Column 2: x coordinate of the reference sensor
Column 3: y coordinate of the reference sensor
Column 4: z coordinate of the reference sensor
Column 5: direction of the measurement
1 - linear along the x-direction
2 - linear along the y-direction
3 - linear along the z-direction
4 - rotation around the x-direction
5 - rotation around the y-direction
6 - rotation around the z-direction

ERRSENS.ext

Specifies the error sensor configuration.

Line 1 :
Column 1 : Number of error sensors (Ns),
Columns 2 to 5 : 0,

Line 2 to (Ns+1) :

Column 1 : Node number of the center of mass of the engine if the error sensor is placed on the engine, or node number of the center of mass of the raft if the error sensor is placed on the raft.

Column 2: x coordinate of the error sensor

Column 3: y coordinate of the error sensor

Column 4: z coordinate of the error sensor

Column 5: direction of the measurement

1 - linear along the x-direction
2 - linear along the y-direction
3 - linear along the z-direction
4 - rotation around the x-direction
5 - rotation around the y-direction
6 - rotation around the z-direction

MONSENS.ext

Specifies the monitoring sensor configuration.

Line 1 :

Column 1 : Number of monitoring sensors (Nm),

Columns 2 to 5 : 0,

Line 2 to (Nm+1) :

Column 1 : Node number of the center of mass of the engine if the monitoring sensor is placed on the engine, or node number of the center of mass of the raft if the error sensor is placed on the raft.

Column 2: x coordinate of the monitoring sensor

Column 3: y coordinate of the monitoring sensor

Column 4: z coordinate of the monitoring sensor

Column 5: direction of the measurement

1 - linear along the x-direction
2 - linear along the y-direction
3 - linear along the z-direction
4 - rotation around the x-direction
5 - rotation around the y-direction
6 - rotation around the z-direction

4. Examples

In this section example data files are given for each of the programs.

4.1 VVESANALYSIS -Engine Supported on two layers of Isolators with a Rigid Raft

In this example a diesel engine is assumed to be supported by six isolators, which themselves are supported on a rigid raft that is supported by four isolators at its corner points. The first six isolators are made of CR and the other four isolators are made of EAR. All isolators have the same size: .2m x .4m x .2m. The geometry of the problem is shown in Figure 1. The units used throughout this example are Newtons, meters and kilograms.

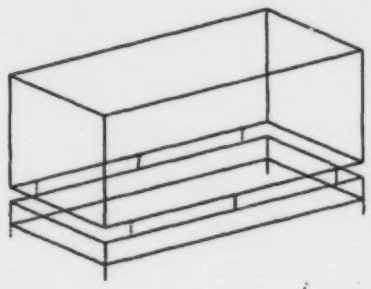


Figure 1 Geometry of Engine Support System

Data Files for this Example

NODE.ext

```
22      0      0      0
 0     -.6     0      1
  .25  -.6     0.3    2
 1.25  -.6     0.3    3
 2.25  -.6     0.3    4
 2.50  -.6     0      5
 2.5    .6     0      6
 2.25   .6     0.3    7
```

```
1.25     .6     0.3     8
 .25     .6     0.3     9
0        .6     0      10
 .25    -.6      .5    11
1.25    -.6      .5    12
2.25    -.6      .5    13
2.25     .6      .5    14
1.25     .6      .5    15
 .25     .6      .5    16
1.25    0       1.25   17
0       -.6     -.2    18
2.50    -.6     -.2    19
2.5      .6     -.2    20
0        .6     -.2    21
1.25    0       0.15   22
```

%matrix defining nodal coordinates
%The first row:
%Column 1: NNODE; Column 2 to Column 4: 0 0 0
%The other rows:
%Column 1: x coordinate
%Column 2: y coordinate
%Column 3: z coordinate
%Column 4: node number

NXYZ.ext

```
2  2  2
2  2  2
```

For rectilinear solid: Column 1: number of elements in the x-direction
Column 2: number of elements in the y-direction
Column 3: number of elements in the z-direction

For solid and hollow cone:
Column 1: number of elements in the radial direction
Column 2: number of elements in the circumferential direction
Column 3: number of elements in the z-direction

ENGINE.ext

```
5850 1800 3750 4100 1.2 2.5 1.5 17  11  12  13  14  15  16
```

% Definition of engine parameters. NSP = number of engine support points:
%Column 1: Mass
%Column 2: mass moment of inertia about X axis
% Column 3: mass moment of inertia about Y axis

% Column 4: mass moment of inertia about Z axis
% Column 5: length of engine in the X-direction [only used for graphics]
% Column 6: length of engine in the Y-direction [only used for graphics]
% Column 7: length of engine in the Z-direction [only used for graphics]
% Column 8: nodal number of center of mass of the engine
% Column 9: engine support node number 1
% Column 10: engine support node number 2
% .
% .
% .
% Column 8+NSP: engine support node number NSP

RAFT.ext

```
2000 125 379 504 2.5 1.2 22 2 3 4 7 8 9  -1  -5  -6  -10
```

NR is the number of raft support points. NR=NRT+NRB where NRT= number of isolators connected to the top of the raft and NRB= number connected to the bottom of the raft. :

Column 1: M (kg). (If the flexible raft option is required, set this number to zero)
Column 2: I_{xx} mass moment of inertia about X axis (kg-m^2)
Column 3: I_{yy} mass moment of inertia about Y axis (kg-m^2)
Column 4: I_{zz} mass moment of inertia about Z axis (kg-m^2)
Column 5: length of raft in the X-direction (only used for graphics)
Column 6: length of raft in the Y-direction (only used for graphics)
Column 7: node number of center of mass of the raft
Column 8: node number of connection 1 between top of the raft and engine
Column 9: node number of connection 2 between top of the raft and the engine on the raft top
.....
Column 7+NRT: node number of connection NRT between top of the raft and the engine
Column 7+NRT+1: minus node number of connection 1 between underside of the raft and the ground
Column 7+NRT+2: minus node number of connection 2 between underside of the raft and the ground
.....
Column 7+NRT+NRB: minus node number of connection NRB between underside of the raft and the ground

ELEMENT.ext

```
10 0  0    0   0   0
1  1  2   11   3   1
1  1  3   12   4   2
1  1  4   13   3   3
1  1  7   14   6   4
1  1  8   15   6   5
1  1  9   16   6   6
1  2 18    1  19   7
1  2 19    5  18   8
1  2 20    6  21   9
1  2 21   10  20  10
```

%The first line:
%Column 1: NELEM; Column 2 to Column 6: 0 0 0 0 0 0 0
%The other lines:
%Column 1: type of element
% 1 - for viscoelastic isolator: element matrices computed internally
% 2 - for viscoelastic isolator: element matrices supplied by user
% 3 - beam element
% 4 - mass element
%Column 2: reference number defining the row in which the required element properties
% are found:
% 1 to N - for isolator elements from VISCO.ext [different row numbers define both
different materials and different material representations]
% 1 to NBEAM - for beam elements from BEAMS.ext
% 1 to NMASS - for mass elements from MASSES.ext
%Column 3: node number 1 of this element/isolator
%Column 4: node number 2 of this element/isolator
%Column 5: node number 3 defining the orientation of this element/isolator, which can
% be a real node of the model or a dummy node. This node together with node
% numbers 1 and 2 define a plane which is the local y-z plane of the element.
%Column 6: global element number

BEAMS.ext

```
2.0E11  .3  7850  7.6e-3  18e-6  129e-6  .396e-6
```

% Properties of NBEAM elastic beams:
% for each line
% column 1: Young's Modulus
% column 2: Poisson's Ratio
% column 3: Mass Density
% column 4: Transverse beam area

% column 5: Second moment of area about local y-axis
% column 6: Second moment of area about local x-axis
% column 7: Second moment of area about local z-axis
% column 8: Torsional constant of area about local z-axis

CONSTR.ext

```
 0   0   0   0   0   0    1
 0   0   0   0   0   0    2
 0   0   0   0   0   0    3
 0   0   0   0   0   0    4
 0   0   0   0   0   0    5
 0   0   0   0   0   0    6
 0   0   0   0   0   0    7
 0   0   0   0   0   0    8
 0   0   0   0   0   0    9
 0   0   0   0   0   0   10
 1   1   1   1   1   1   11
 1   1   1   1   1   1   12
 1   1   1   1   1   1   13
 1   1   1   1   1   1   14
 1   1   1   1   1   1   15
 1   1   1   1   1   1   16
 0   0   0   0   0   0   17
-1  -1  -1  -1  -1  -1   18
-1  -1  -1  -1  -1  -1   19
-1  -1  -1  -1  -1  -1   20
-1  -1  -1  -1  -1  -1   21
 0   0   0   0   0   0   22
```

% Definition of nodal constraints
% For each row:
%Column 1:x direction translation =0 free =1 fixed =-1 as a supported DOF
%Column 2:y direction translation =0 free =1 fixed =-1 as a supported DOF
%Column 3:z direction translation =0 free =1 fixed =-1 as a supported DOF
%Column 4:x direction rotation =0 free =1 fixed =-1 as a supported DOF
%Column 5:y direction rotation =0 free =1 fixed =-1 as a supported DOF
%Column 6:z direction rotation =0 free =1 fixed =-1 as a supported DOF
%Column 7:node number

VISCO.EXT

```
1 0.688e7 28 139  3412  6445  1340  0.4995  68.8e8 20  3000
1 0.642e8 38 348  6048  7041  1280  0.4993  458.e8 30  4500
```

VISCO.ext (1:N, 1:11) matrix defining the isolator viscoelastic properties. The total

number is N and each row corresponds to an isolator property defined in COORD.ext.

i) for viscoelastic characteristics of the isolators defined in terms of the rational polynomial (RP) formula. For complex Poisson's ratio case, there are two complex moduli

$$G_1^*(i\omega) = G_1(1 - \sum_{n=1}^{2} \frac{a_{1n}}{i\omega + b_n})$$

$$G_2^*(i\omega) = G_2(1 - \sum_{n=1}^{2} \frac{a_{2n}}{i\omega + b_n})$$

Column 1: 1 (for RP model)
Column 2: G_1 (N/m^2)
Column 3: a_{11} (Hz)
Column 4: b_1 (Hz)
Column 5: a_{12} (Hz)
Column 6: b_2 (Hz)
Column 7: mass density of isolator (kg/m^3)
Column 8: Poisson's Ratio at zero frequency point
Column 9: G_2 (N/m^2)
Column 10: a_{21} (Hz)
Column 11: a_{22} (Hz)

COORD.ext

```
 1   0   0        1   0   0
.2  .4   0       .2  .4   0
.2  .4  .2       .2  .4  .2
 0   0   0        0   0   0
 0   0  .2        0   0  .2
```

% Coordinates defining the geometry of the isolators. There are N types
% of different isolators.
% For every three columns:
% Row 1: Index of the type isolator geometry
% Column 1: =1 Rectangular solid (a: side length for x-direction;
% b: side length for y-direction; L: height in z-direction)
% =2 Hollow cylinder (d: inner diameter; D: outer diameter; L: height in z-direction)
% =3, Solid cylinder (D: diameter; L: height in z-direction)
% Row 2:
% Column 1: a1 for index=1; d1 for index=2; 0 for index=3
% Column 2: b1 for index=1; D1 for index=2 or 3
% Column 3: z-coordinate of bottom surface of isolator
% Row 3:

% Column 1: a2 for index=1; d2 for index=2; 0 for index=3
% Column 2: b2 for index=1; D2 for index=2 or 3
% (a1,b1,d1,D1 can be different from a2, b2, d2, D2, respectively)
% Column 3: z-coordinate of top surface of isolator
% Row 4: coordinates of base of condensed isolator model
% Row 5: coordinates of top of condensed isolator model

FORCE.ext

```
17   0   0    0   0   0  1  0  0  0
```

%matrix defining the node and its degree of freedom that is loaded for frequency response calculation:
%Column 1: the node loaded or "0" if another load point on engine is required
%Column 2: x coordinate of the applied force, or "0" if applied at any node
%Column 3: y coordinate of the applied force, or "0" if applied at any node
%Column 4: z coordinate of the applied force, or "0" if applied at any node
%Column 5: force along the x-direction =1 if loaded, = 0 otherwise
%Column 6: force along the y-direction =1 if loaded, = 0 otherwise
%Column 7: force along the z-direction =1 if loaded, = 0 otherwise
%Column 8: moment around the x-direction =1 if loaded, = 0 otherwise
%Column 9: moment around the y-direction =1 if loaded, = 0 otherwise
%Column10: moment around the z-direction =1 if loaded, = 0 otherwise

RESP.ext

```
0  60  100   17  0   0
```

% vector defining frequency range and the node at which the frequency response characteristic is required:
%Column 1: the lowest frequency (Hz) for frequency response calculation
%Column 2: the highest frequency (Hz) for frequency response calculation
%Column 3: the number of discrete frequency points
%Column 4: node number for which the frequency response calculation is required
%Column 5: node number n1
%Column 6: node number n2

FTRANS.ext

```
18   0   0    1.25   .5  0  60  100
```

% Definition of force transmissibility parameters. Force can be applied at C of G or
% any other point on the engine:
%Column 1: support node number at which the transmissibility is calculated

%Column 2: the node number of the centre of mass or "0" if another load point is
% required
%Column 3: x coordinate of the applied force, or "0" if applied at centre of mass
%Column 4: y coordinate of the applied force, or "0" if applied at centre of mass
%Column 5: z coordinate of the applied force, or "0" if applied at centre of mass
%Column 6: the lowest frequency (Hz) for frequency response calculation
%Column 7: the highest frequency (Hz) for frequency response calculation
%Column 8: the number of discrete frequency points used in the analysis

TFTRANS.ext

```
0     1     .25    .5   0   60   100
```

% Definition of average force transmissibility parameters. Force can be applied at C of G
% or any other point on the engine.
%Column 1: the node number of centroid or "0" if another load point is required
%Column 2: x coordinate of the applied force, or "0" if applied at centre of mass
%Column 3: y coordinate of the applied force, or "0" if applied at centre of mass
%Column 4: z coordinate of the applied force, or "0" if applied at centre of mass
%Column 5: the lowest frequency (Hz) for frequency response calculation
%Column 6: the highest frequency (Hz) for frequency response calculation
%Column 7: the number of discrete frequency points used in the analysis

DTRANS.ext

```
4    18    19    20    21   17   -10    0    0.95    0    5    40
```

% Definition of displacement transmissibility functions:
% BNOD = number of nodes rigidly connected to base structure.
%Column 1: BNOD
%Column 2: base node number 1
% .
% .
%Column BNOD+1: base node number BNOD
%Column BNOD+2: node number at which displacement
% transmissibility functions are to be calculated
%Column BNOD+3: x coordinate of the origin of the support motion
%Column BNOD+4: y coordinate of the origin of the support motion
%Column BNOD+5: z coordinate of the origin of the support motion
%Column BNOD+6: the lowest frequency (Hz) for calculation
%Column BNOD+7: the highest frequency (Hz) for calculation
%Column BNOD+8: the number of discrete frequency points

Results of Free Vibration Analysis:

NATURAL FREQUENCIES AND MODE SHAPES:

Order	Frequency (Hz)	Mode Shape
1	-0.0222+ i1.9963	Engine Ty motion (Translation in Y direction. Engine and raft move in the same direction
2	-0.0301+ i2.3926	Engine Tx motion (translation in X direction). Engine and raft move in the same direction
3	-0.0716+ i3.9295	Engine Rz rotation (rotation about Z direction). Engine and raft rotate in the same direction
4	-0.1710+ i5.6062	Engine Tz motion (bounce mode). Engine and raft move in the same direction
5	-0.2678+ i6.7667	Engine Rx rotation. Engine and raft move in the same direction
6	-0.2555+ i7.1669	Engine Ry rotation. Engine and raft move in the same direction
7	-0.4758+ i11.5440	Tx motion. Engine and raft move in the opposite Direction
8	-0.5359+ i12.6625	Ty motion. Engine and raft move in the opposite Direction
9	-1.9145+ i21.5426	Raft Rz rotation. Engine stationary
10	-2.3445+ i21.6357	Raft Tz motion. Engine almost stationary
11	-10.5198+ i44.3284	Raft Ry rotation. Engine stationary
12	-9.4424+ i46.3881	Raft Rx rotation. Engine stationary

In the above listing the first number is the decay coefficient which arises as a result of the viscoelastic characteristic of the problem and the second number is the natural frequency in Hz. Figure A1 shows the first mode shape.

Figure A2 shows the velocity frequency response function for force excitation applied at the centre of mass (node 17) in the Y direction and velocity response measured in the Z and Y directions for the same node. In this case, because of the assumed symmetry of the engine and its supports, there is no response in the Z direction. The resonant peak occurs at 1.8 Hz corresponding to the natural frequency at this point.

Figures A3 and A4 show the force transmissibility function between a point on the engine with coordinates (1.0, .25, .5) and the forces applied to support node 18 in the Y and Z directions.

Figures A5 and A6 show the displacement transmissibility function between the mass center of engine and the base in the Y and Z directions.

Figure A7, A8 and A9 show the total force transmissibility function between a point on the engine with coordinates (1.0, .25, .5) and the forces applied to base.

Mode Shape Number 1: Complex Frequency= -0.022230 +i 1.996311 (Hz)

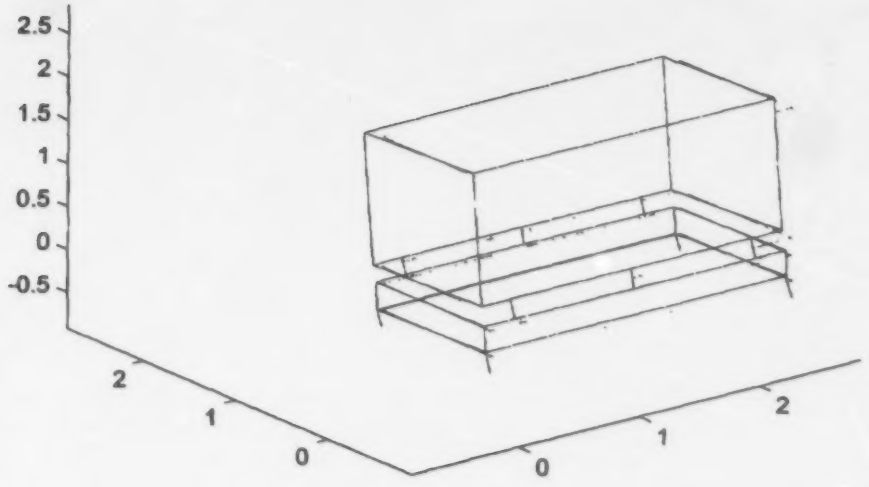


Figure A1 The First Mode Shape

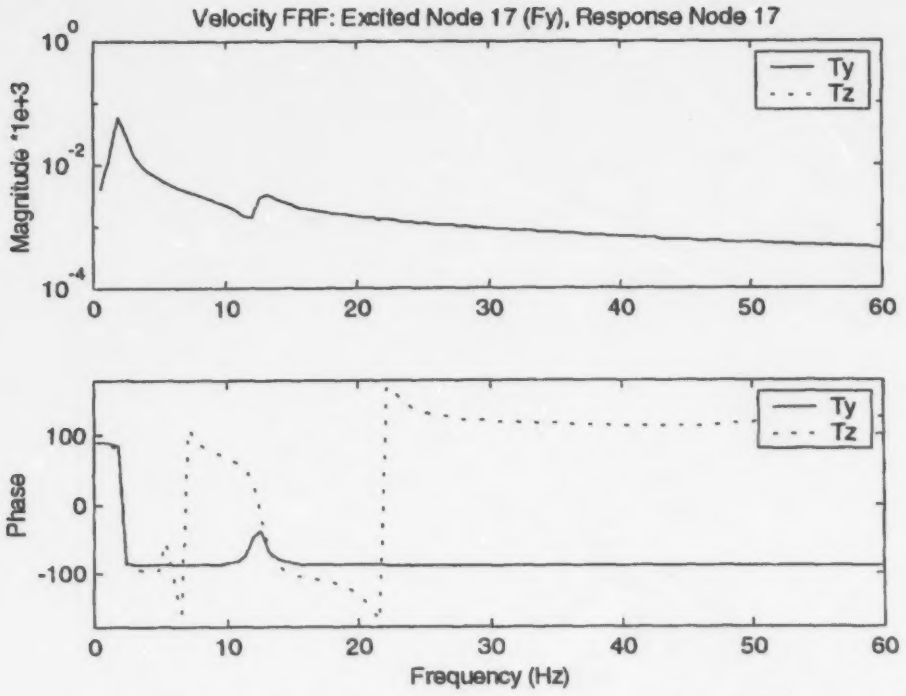


Figure A2 Velocity Frequency Response Functions

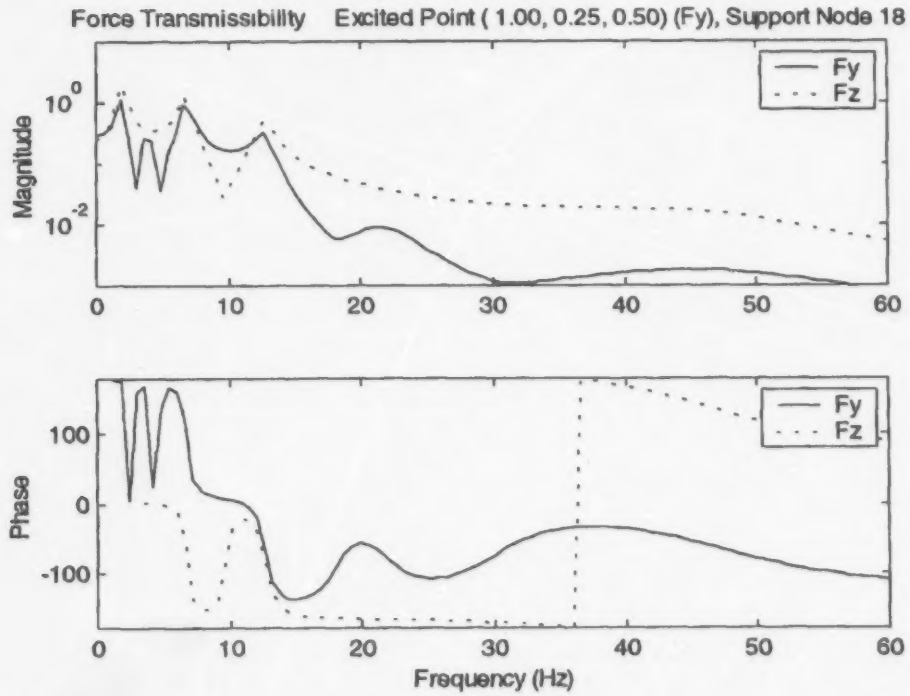


Figure A3 Force Transmissibility Functions

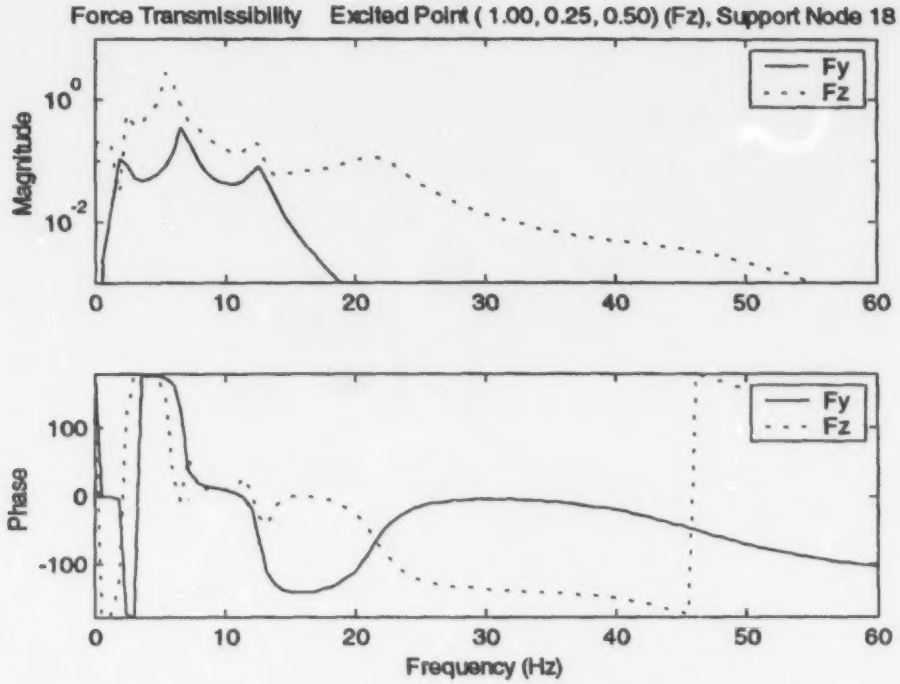


Figure A4 Force Transmissibility Functions

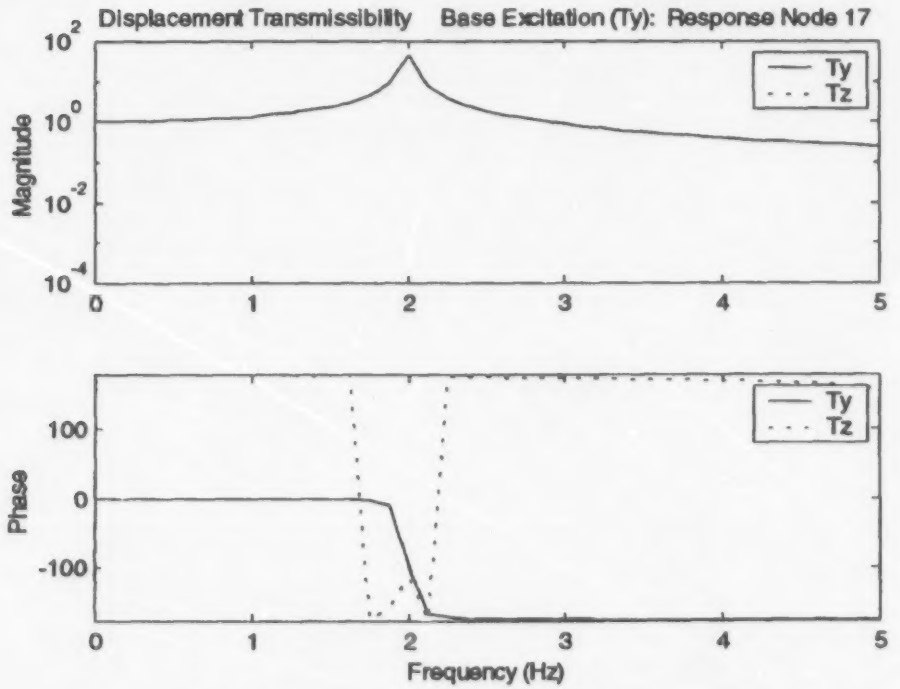


Figure A5 Displacement Transmissibility Functions

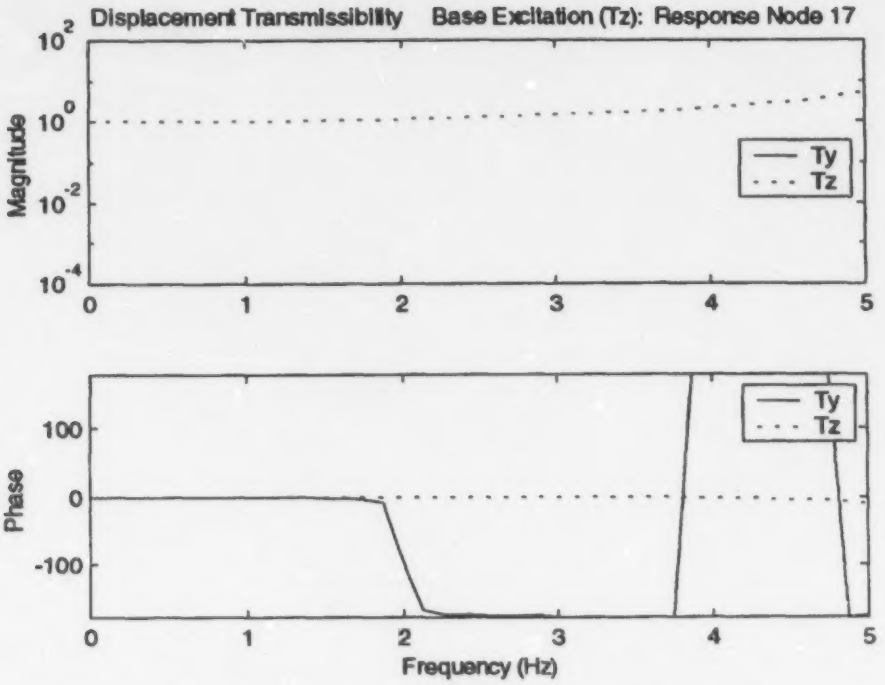


Figure A6 Displacement Transmissibility Functions

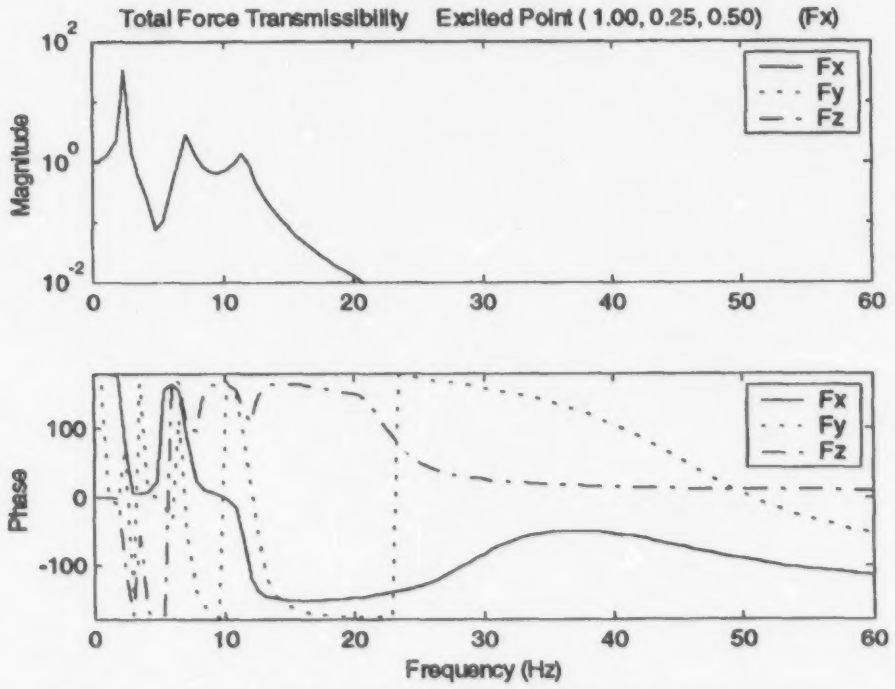


Figure A7 Total Force Transmissibility Functions

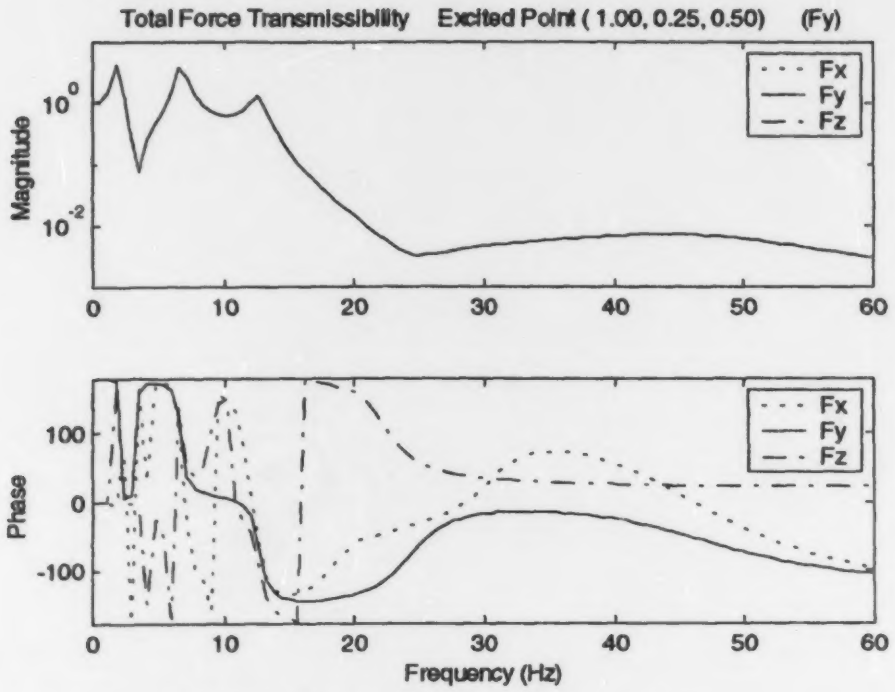


Figure A8 Total Force Transmissibility Functions

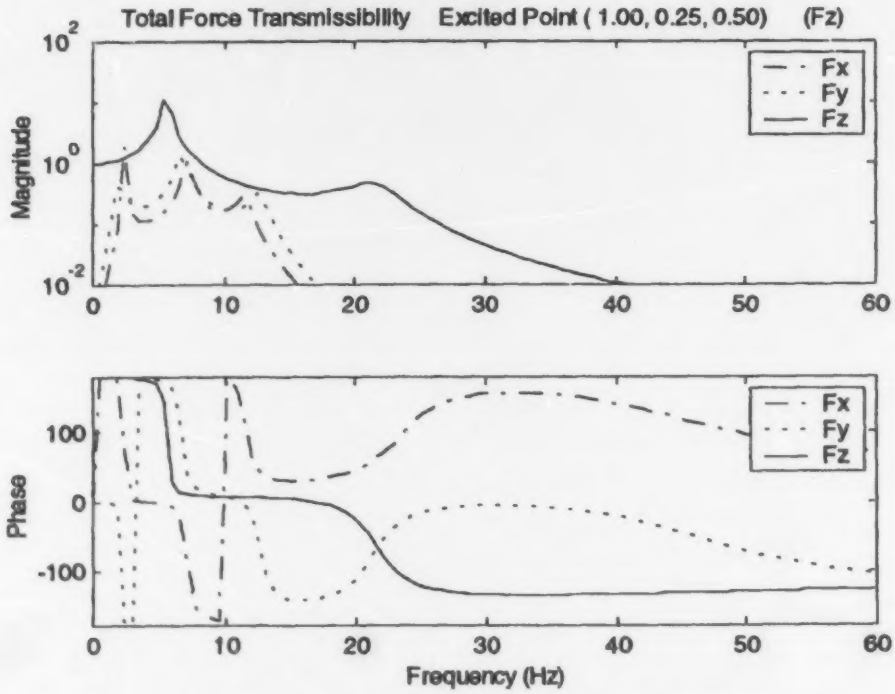


Figure A9 Total Force Transmissibility Functions

4.2 VVESDYNSTFF -Engine Supported on two layers of Isolators with a Rigid Raft.

This is the same example as presented in section 4.1 except that the input data for isolator is now to use direct stiffness input. The data files that are common to the example in section 4.1 are not repeated.

VISCODS.ext

```
1  0   0 0  0   0   1340  0.4995 0 0 0
1  0   0 0  0   0   1280  0.4993 0 0 0
```

Matrix defining the viscoelastic properties of the isolator
the first column defines the way in which the complex Young's modulus is defined
column 1=1: rational polynomial definition
column 1=2: Havrilak Negami model definition
column 1=3: experimental data provided in DVISCO.ext
For definition of other columns see the user's manual

DIRECTSTIFF.ext

Matrix defining the measured stiffness characteristics of the isolators. In this case these have been calculated internally and the results are not repeated here because of the size of the data files. They can be found with the program files.

Results of Free Vibration Analysis:

NATURAL FREQUENCIES AND MODE SHAPES:

Order	Frequency (Hz)	Mode Shape
1	-0.0222+ i1.9965	Engine Ty motion (Translation in Y direction. Engine and raft move in the same direction
2	-0.0301+ i2.3929	Engine Tx motion (translation in X direction). Engine and raft move in the same direction
3	-0.0715+ i3.9404	Engine Rz rotation (rotation about Z direction). Engine and raft rotate in the same direction
4	-0.1706+ i5.6101	Engine Tz motion (bounce mode). Engine and raft move in the same direction
5	-0.2670+ i6.7758	Engine Rx rotation. Engine and raft move in the same direction
6	-0.2546+ i7.1738	Engine Ry rotation. Engine and raft move in the same direction
7	-0.4720+ i11.5629	Tx motion. Engine and raft move in the opposite Direction
		Ty motion. Engine and raft move in the opposite

8	-0.5307+ i12.6842	Direction
9	-1.8627+ i21.7059	Raft Rz rotation. Engine stationary
10	-2.2790+ i21.8775	Raft Tz motion. Engine almost stationary
11	-9.4027+ i46.3152	Raft Ry rotation. Engine stationary
12	-8.4151+ i47.8204	Raft Rx rotation. Engine stationary

In the above listing it can be seen the first number is the decay coefficient which arises as a result of the viscoelastic characteristic of the problem and the second number is the natural frequency in Hz. Figure A10 shows the first mode shape. .

Figure A11 shows the velocity frequency response function for force excitation applied at the centre of mass (node 17) in the Y direction and velocity response measured in the Z and Y directions for the same node. In this case, because of the assumed symmetry of the engine and its supports, there is no response in the Z direction. The resonant peak occurs at 1.8 Hz corresponding to the natural frequency at this point.

Figures A12 and A13 show the force transmissibility function between a point on the engine with coordinates (1.0, .25, .5) and the forces applied to support node 18 in the Y and Z directions.

Figures A14 and A15 show the displacement transmissibility function between the mass center of engine and the base.

Figures A16, A17 and A18 show the total force transmissibility function between a point on the engine with coordinates (1.0, .25, .5) and the forces applied to base.

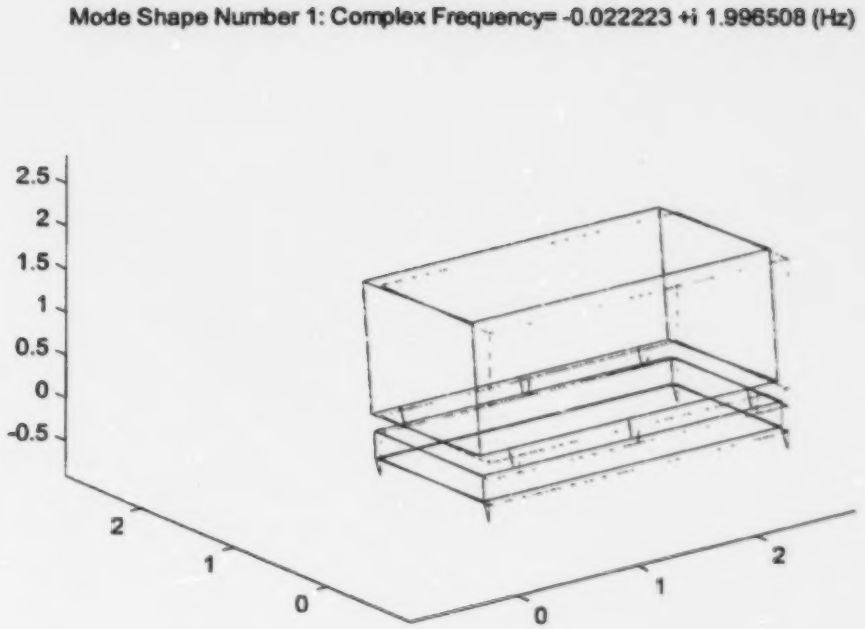


Figure A10 The First Mode Shape

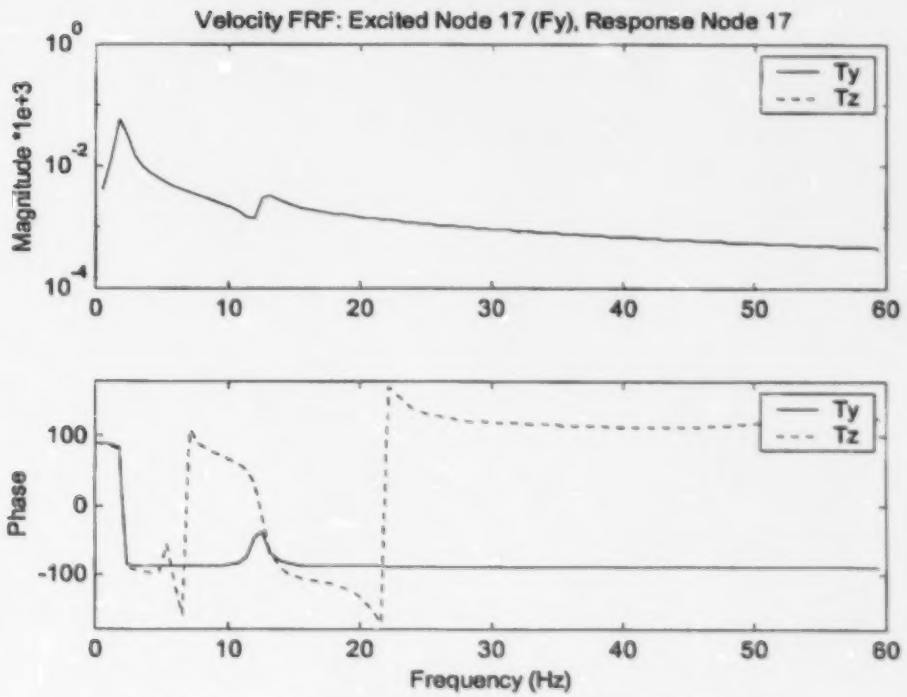


Figure A11 Velocity Frequency Response Functions

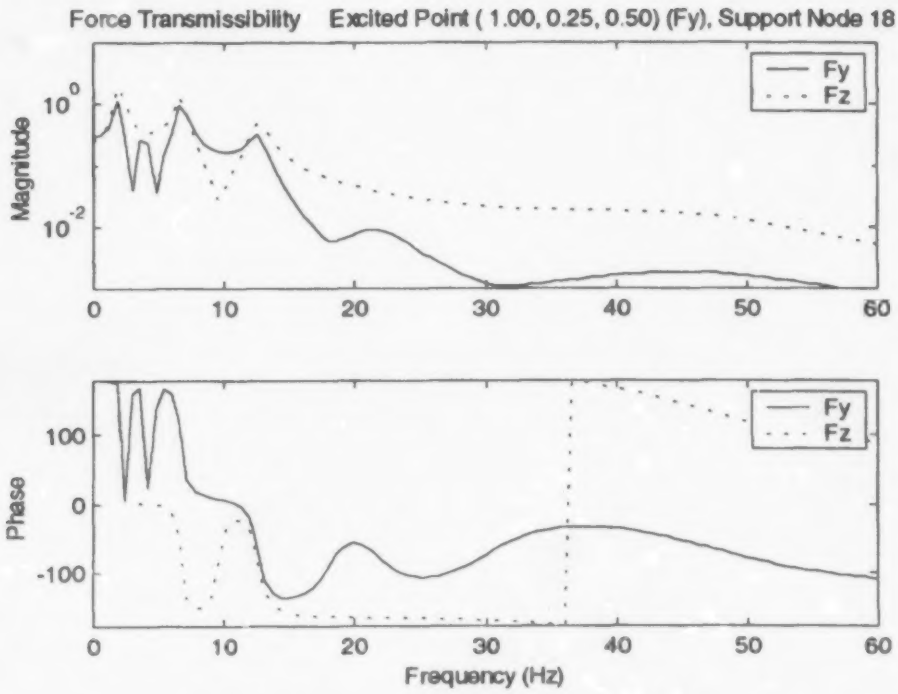


Figure A12 Force Transmissibility Functions

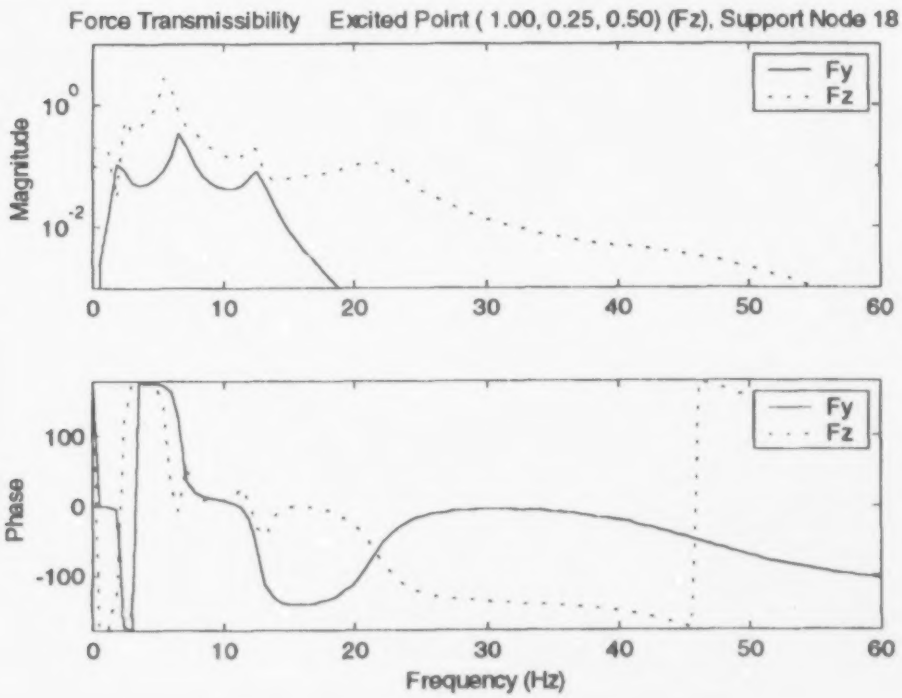


Figure A13 Force Transmissibility Functions

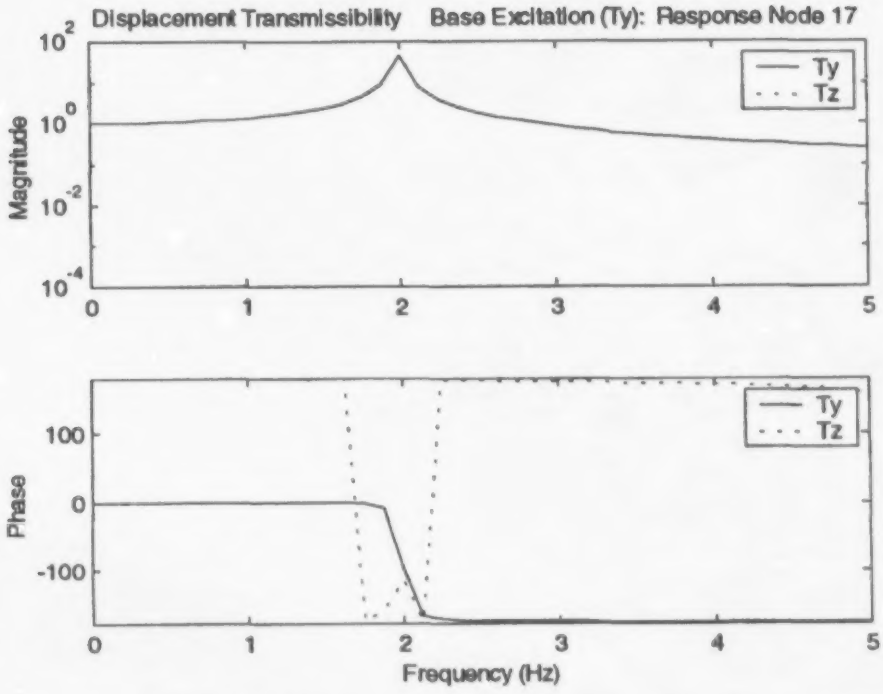


Figure A14 Displacement Transmissibility Functions

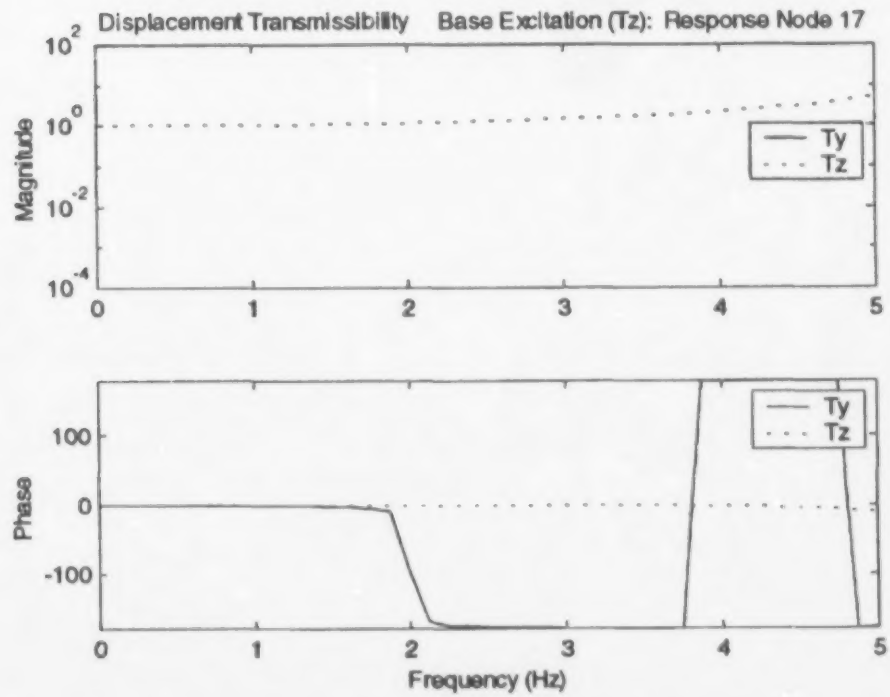


Figure A15 Displacement Transmissibility Functions

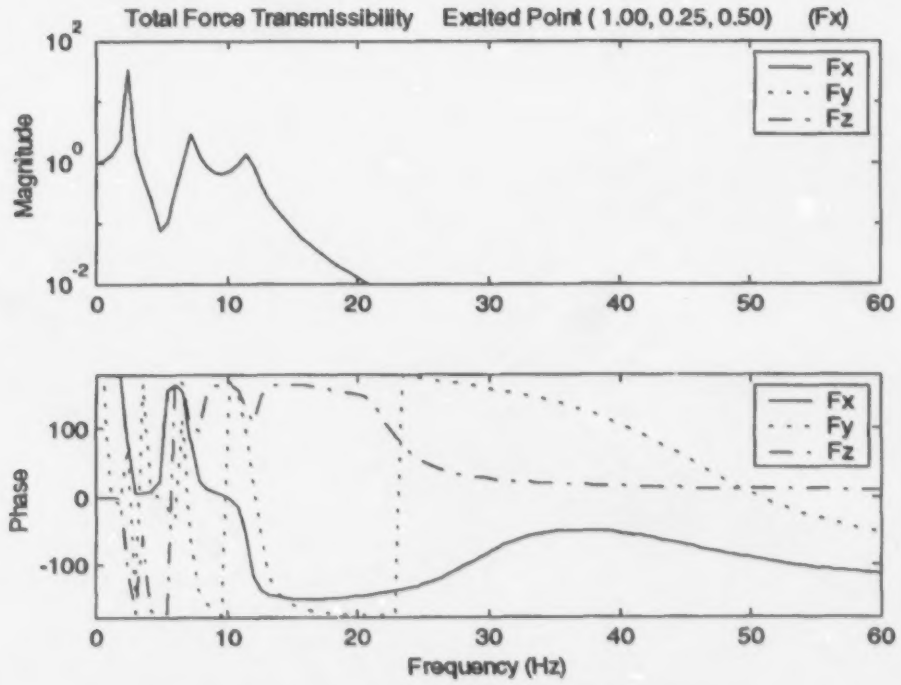


Figure A16 Total Force Transmissibility Functions

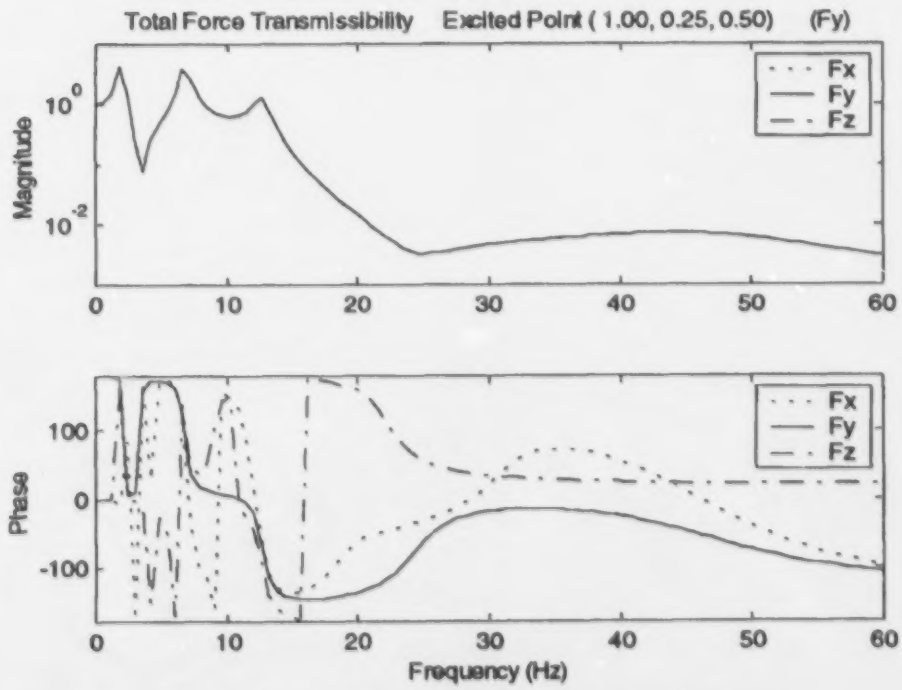


Figure A17 Total Force Transmissibility Functions

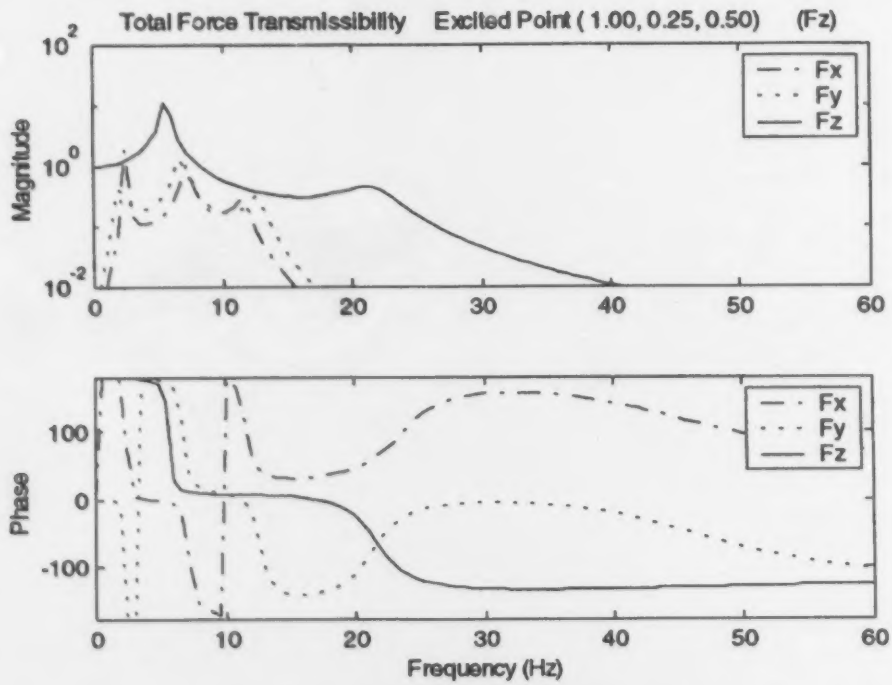


Figure A18 Total Force Transmissibility Functions

4.3 VVES4POLE.m -Engine Supported on two layers of Isolators with a Rigid Raft.

This is a similar example as presented in section 4.1 except that the input data for isolator is input in the form of transfer function (four-pole data) data. In this case the input data is calculated from the same parameters that were used in example 4.1. Results are presented to compare the results obtained between the case where the data input consists of only three sets of four-pole data (case 2) and the case (case 3) where this four-pole data is supplemented by analytical data.

All those data files that are common to the example in section 4.1 are not repeated. OFHZ.ext, fpx1.ext, fpy1.ext, fpz1.ext, fpx2.ext, fpy2.ext and fpz2.ext are not listed because of their size.

NODE.ext

```
23      0       0       0
 0      -.6     0       1
  .25   -.6     0.3     2
 1.25   -.6     0.3     3
 2.25   -.6     0.3     4
 2.50   -.6     0       5
 2.5     .6     0       6
 2.25    .6     0.3     7
 1.25    .6     0.3     8
```

```
  .25     .6     0.3     9
   0      .6     0      10
  .25    -.6      .5    11
 1.25    -.6      .5    12
 2.25    -.6      .5    13
 2.25     .6      .5    14
 1.25     .6      .5    15
  .25     .6      .5    16
 1.25      0     1.25   17
 0       -.6     -.2    18
 2.50    -.6     -.2    19
 2.5      .6     -.2    20
   0      .6     -.2    21
 1.25    0       0.15   22
 1.25    0       -.2    23
```

%matrix defining nodal coordinates
%The first row:
%Column 1: NNODE; Column 2 to Column 4: 0 0 0
%The other rows:
%Column 1: x coordinate
%Column 2: y coordinate
%Column 3: z coordinate
%Column 4: node number

ELEMENT4.ext

```
10 0  0    0  0    0   0    0
1  1 18    1 19   23  22    1
1  1 19    5 18   23  22    2
1  1 20    6 21   23  22    3
1  1 21   10 20   23  22    4
1  2  2   11  3   22  17    5
1  2  3   12  4   22  17    6
1  2  4   13  3   22  17    7
1  2  7   14  6   22  17    8
1  2  8   15  6   22  17    9
1  2  9   16  6   22  17   10
```

%The first line:
%Column 1: NELEM; Column 2 to Column 8: 0 0 0 0 0 0 0
%The other lines:
%Column 1: type of element
% 1 - for viscoelastic isolator: element matrices computed internally
% 2 - for viscoelastic isolator: element matrices supplied by user
%Column 2: reference number defining the row in which the required element properties
% are found:
% 1 to N - for isolator elements from VISCO.ext [different row numbers define both different materials and diferent material representations]

of this element/isolator
%Column 4: node number 2 of this element/isolator
%Column 5: node number 3 defining the orientation of this element/isolator, which can be
% a real node of the model or a dummy node. This node together with node
% numbers 1 and 2 define a plane which is the local y-z plane of the element.
%Column 6: node number 4 defining the node rigidly connected with node number 1
%Column 7: node number 5 defining the node rigidly connected with node number 2
%Column 8: global element number

NFREQ.ext

```
1  21  101
```

%Column 1: the lowest frequency (Hz) for calculation
%Column 2: the highest frequency (Hz) for calculation
%Column 3: the number of discrete frequency points used in the analysis

Figure A19 shows the magnitude of its diagonal force transmissibility functions (its diagonal velocity transmissibility functions are the same). Figure A20 shows the magnitude of its frequency response functions.
Figures A21-32 show the comparisons between the results calculated from case 1 and case 2. In case 2 the only isolator data used is the 4pole data for the x, y and z directions. In case 1 the complete transmissibility matrix is used. Case 1 is denoted by the label c and case 2 by the label s. The results show that in general the limited experimental data predicts the almost the same response as the full set of data. The results show for motion in the vertical direction the two analyses give exactly the same results. This is because for this case the actual deformation of the isolators is exactly the same as in the test. It is not clear why the torsional response given in Figure A26 shows such close agreement. In general the extent of agreement will depend upon the similarity between the actual and tested responses of the isolator.

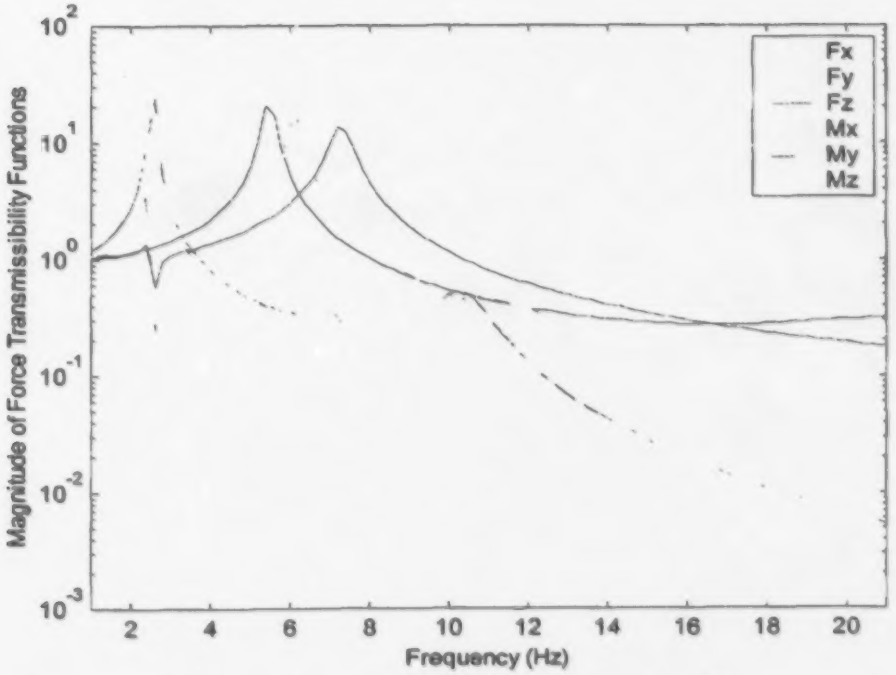


Figure A19. Force Transmissibility Functions

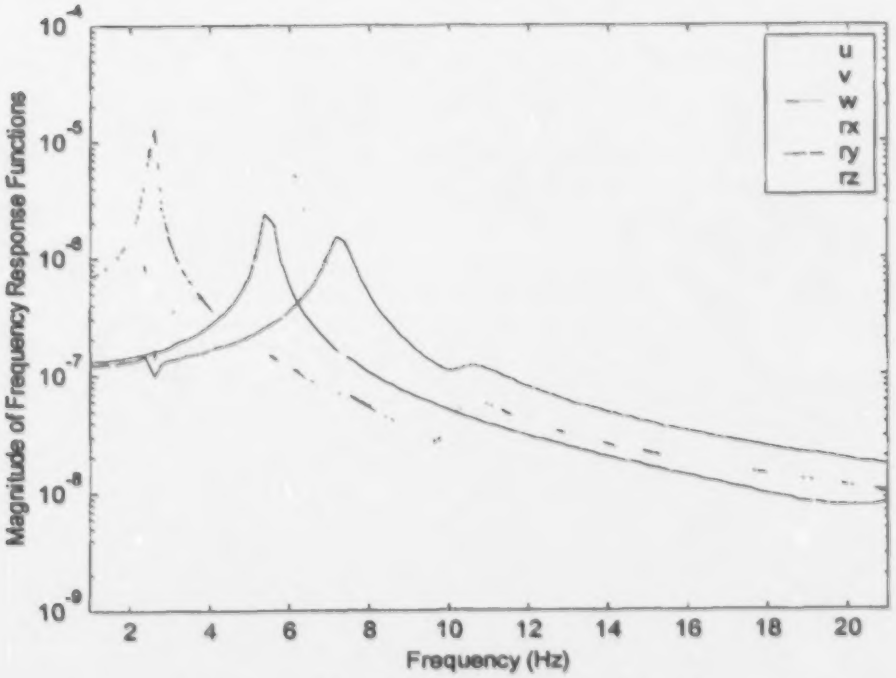


Figure A20. Frequency Response Functions

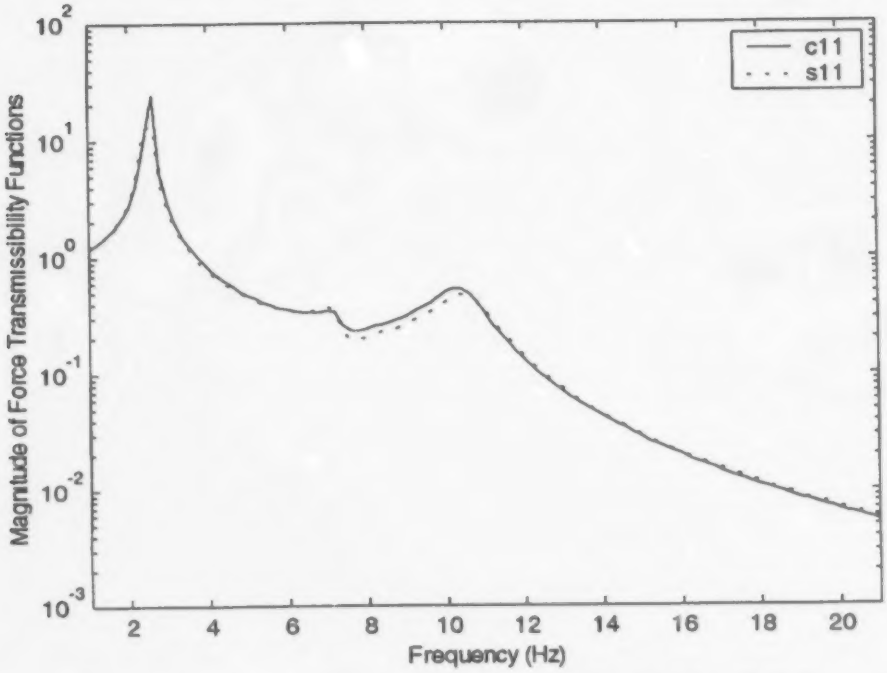


Figure A21. Force Transmissibility Function Matrix: X-X Force

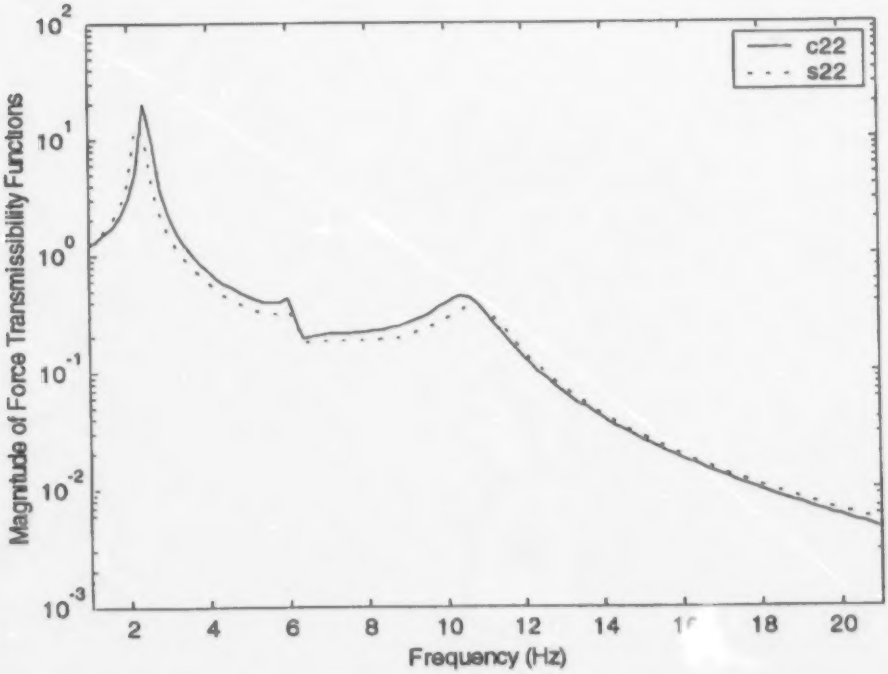


Figure A22. Force Transmissibility Function: Y-Y Force

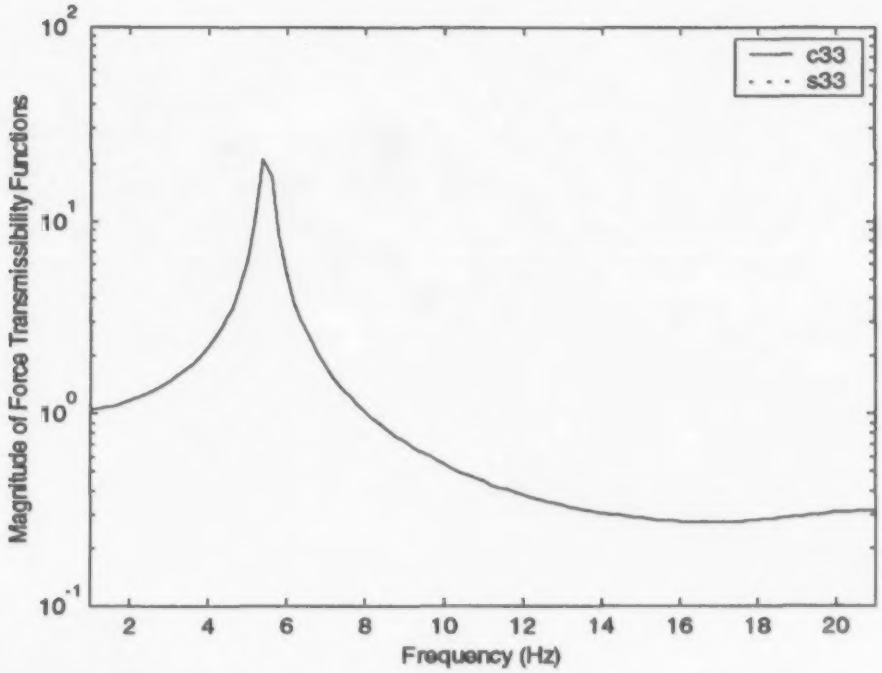


Figure A23. Force Transmissibility Function : Z-Z Force

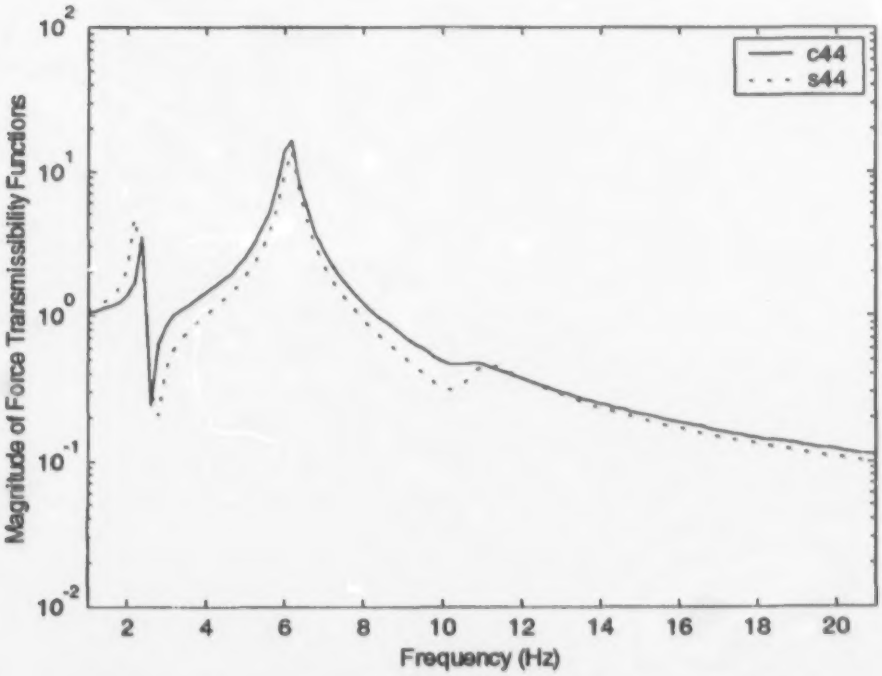


Figure A24. Force Transmissibility Function : X-X Moment

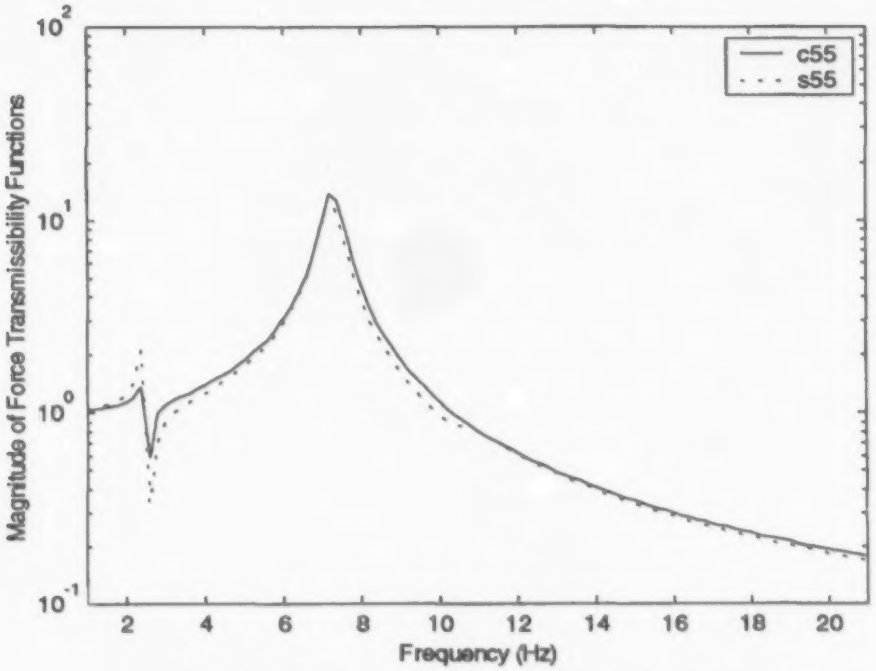


Figure A25. Transmissibility Function : Y-Y Moment

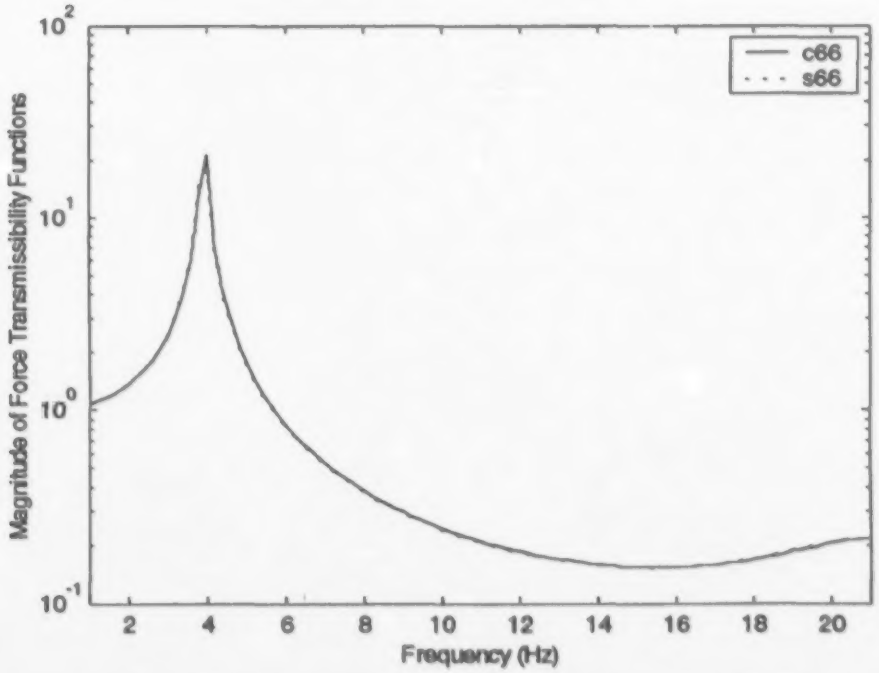


Figure A26. Force Transmissibility Function: Z-Z Moment

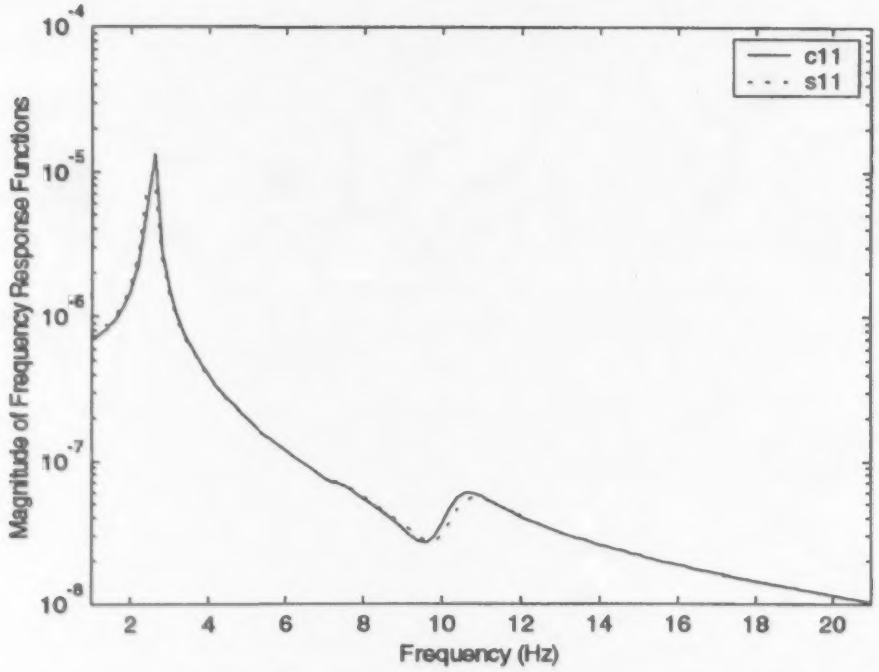


Figure A27. Velocity Frequency Response Function : X-X

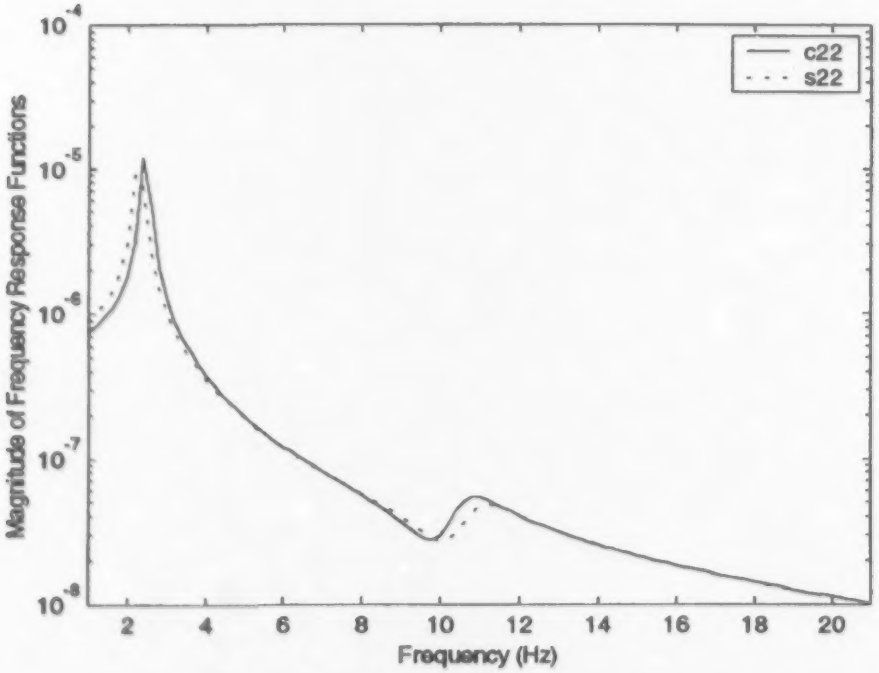


Figure A28. Velocity Frequency Response Function: Y-Y

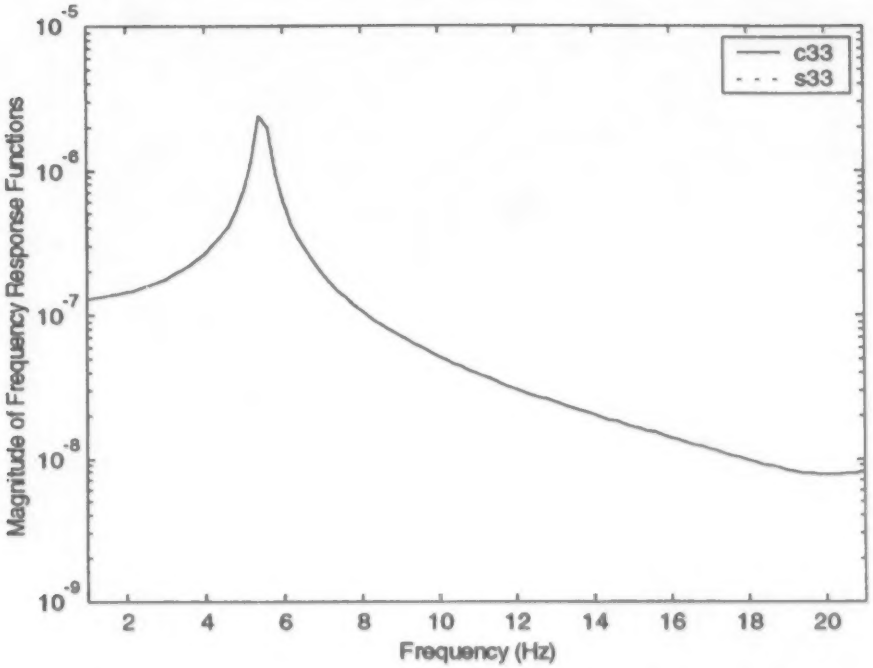


Figure A29. Velocity Frequency Response Function: Z-Z

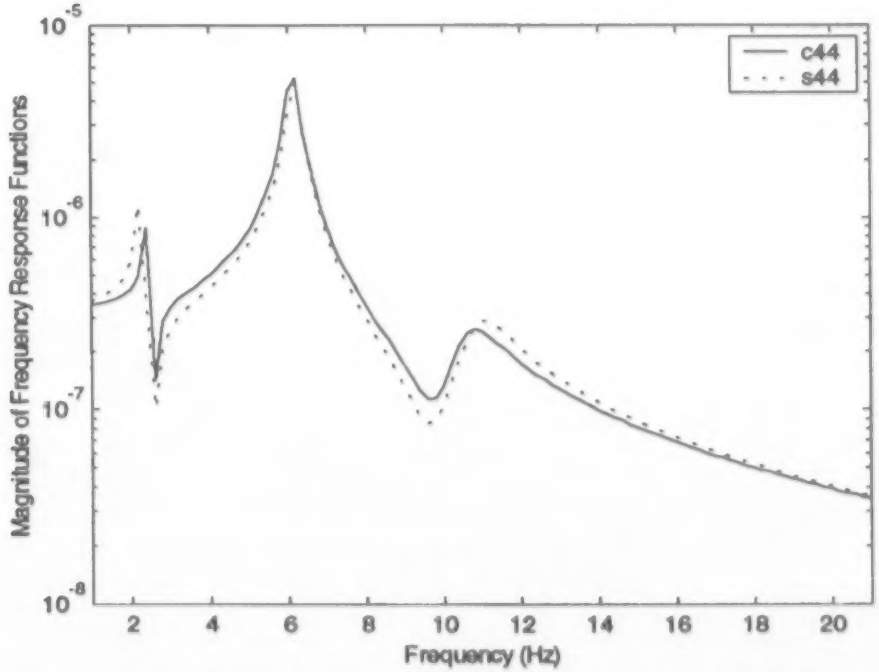


Figure A30. Velocity Frequency Response Function: RX-RX

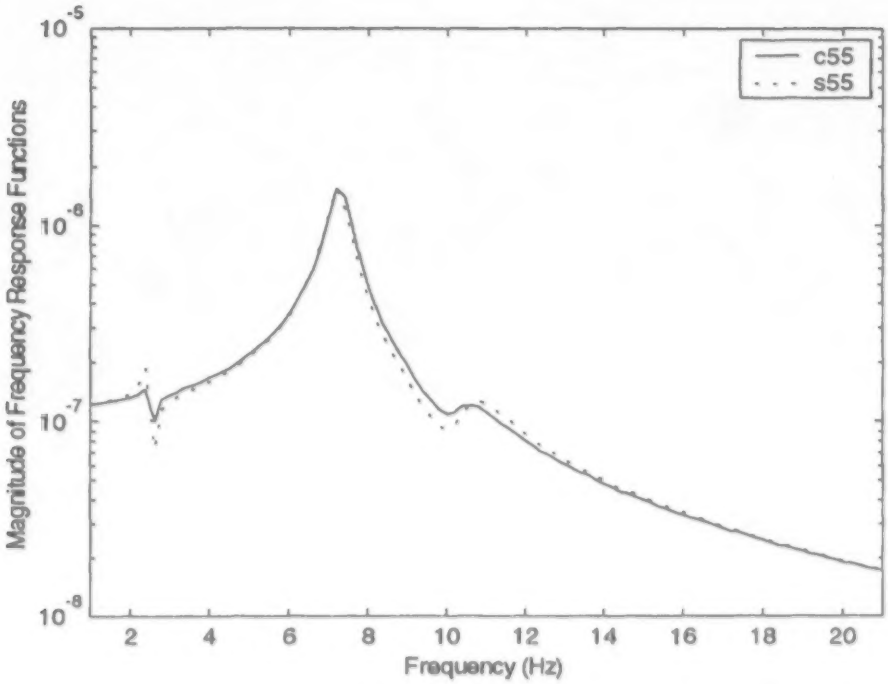


Figure A31. Velocity Frequency Response Function: RY-RY

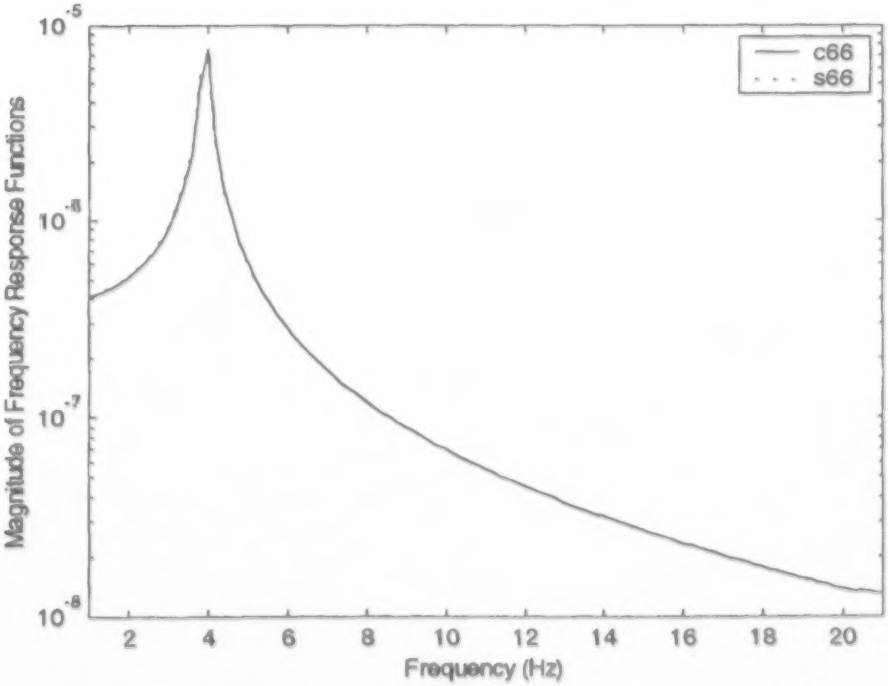


Figure A32. Velocity Frequency Response Function: RZ-RZ

4.4 VVESACTIVE -Engine Supported on a Rigid Raft.

Effect of active control on transmitted vibrations.

This is the same example as presented in section 4.1 In this example a diesel engine is assumed to be supported by six isolators, which themselves are supported on a rigid raft that is supported by four isolators at its corner points. Four control force actuators in the z-direction are inserted in the corners, between the engine and the raft, for the active control of vibration of the engine. Four error sensors are placed in the four lower corners of the raft to measure the displacement in the z-direction. Four disturbance forces (along Ty, Tz, Ry and Rz), composed of two harmonics (5 Hz and 10 Hz) are applied at the center of mass of the engine. All isolators have the same size: .2m x .4m x .2m. The geometry of the problem is shown in Figure A33. The units used throughout this example are Newtons, meters and kilograms.

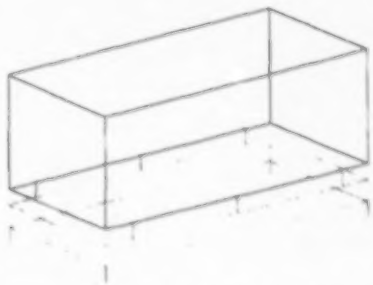


Figure A33: Geometry of engine support system

INPUT DATA FOR VVES_AVC.m:

NODE.EXT

The total number nodes (NNODE)= 22:

0.0000	-0.6000	0.0000	1
0.2500	-0.6000	0.3000	2
1.2500	-0.6000	0.3000	3
2.2500	-0.6000	0.3000	4
2.5000	-0.6000	0.0000	5
2.5000	0.6000	0.0000	6
2.2500	0.6000	0.3000	7
1.2500	0.6000	0.3000	8
0.2500	0.6000	0.3000	9
0.0000	0.6000	0.0000	10
0.2500	-0.6000	0.5000	11
1.2500	-0.6000	0.5000	12
2.2500	-0.6000	0.5000	13
2.2500	0.6000	0.5000	14
1.2500	0.6000	0.5000	15

0.2500	0.6000	0.5000	16
1.2500	0.0000	1.2500	17
0.0000	-0.6000	-0.2000	18
2.5000	-0.6000	-0.2000	19
2.5000	0.6000	-0.2000	20
0.0000	0.6000	-0.2000	21
1.2500	0.0000	0.1500	22

ELEMENT.EXT

The total number of elements (NELEM)= 10:

1	1	2	11	3	1
1	1	3	12	4	2
1	1	4	13	3	3
1	1	7	14	6	4
1	1	8	15	6	5
1	1	9	16	6	6
1	2	18	1	19	7
1	2	19	5	18	8
1	2	20	6	21	9
1	2	21	10	20	10

CONSTR.EXT

1	1	1	1	1	1	1
1	1	1	1	1	1	2
1	1	1	1	1	1	3
1	1	1	1	1	1	4
1	1	1	1	1	1	5
1	1	1	1	1	1	6
1	1	1	1	1	1	7
1	1	1	1	1	1	8
1	1	1	1	1	1	9
1	1	1	1	1	1	10
1	1	1	1	1	1	11
1	1	1	1	1	1	12
1	1	1	1	1	1	13
1	1	1	1	1	1	14
1	1	1	1	1	1	15
1	1	1	1	1	1	16
0	0	0	0	0	0	17
-1	-1	-1	-1	-1	-1	18
-1	-1	-1	-1	-1	-1	19
-1	-1	-1	-1	-1	-1	20
-1	-1	-1	-1	-1	-1	21
0	0	0	0	0	0	22

BEAMS.EXT

2.000000e+011 0.3000 7850.0000 7.600000e-003 1.800000e-005 1.290000e-004
3.960000e-007

MASSES.EXT

0.0000 0.0000 0.0000 0.0000

ENGINE.EXT

5850.0 1800.0 3750.0 4100.0 2.5 1.2 1.5 17 11 12 13 14 15 16

RAFT.EXT

2000.0 125.0 379.0 504.0 2.5 1.2 22 2 3 4 7 8 9 -1
-5 -6 -10

VISCO.EXT

1 6.880000e+006 2.800000e+001 1.390000e+002 3.412000e+003 6.445000e+003
1340.00 0.499 2.000000e+007 2.000000e+001 3.000000e+003
1 6.420000e+007 3.800000e+001 3.480000e+002 6.048000e+003 7.041000e+003
1280.00 0.499 2.000000e+008 3.000000e+001 4.500000e+003

NXYZ.EXT

2 2 2
2 2 2

COORD.EXT

1.0000	0.0000	0.0000	1.0000	0.0000	0.0000
0.2000	0.4000	0.0000	0.2000	0.4000	0.0000
0.2000	0.4000	0.2000	0.2000	0.4000	0.2000
0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
0.0000	0.0000	0.2000	0.0000	0.0000	0.2000

RESP.EXT

0.00 25.00 500 22 0 0

FORCE.EXT

```
 17   0.0000   0.0000   0.0000   0   0   1   0   0   0
```

FTRANS.EXT

```
 18   0     0.00     1.25     0.50     0.00     60.00  100
```

DTRANS.EXT

```
  4   18   19   20   21   17
 -10.00     0.00     0.95     0.00     5.00     40.00
```

QUASI.EXT

```
  30   15
-1.897966e-002 1.833914e+000
```

INPUT DATA FOR AVC1.m

PARAM.EXT

```
100   2500  300  0.00002  100
```

Sampling frequency : 100 Hz,
Number of coefficient for the finite impulse response filters : 2500,
Time length of simulation : 300 s,
Convergence coefficient : 0.00002,
Number of coefficients for the control filters : 100.

DISTURB.EXT

```
4  5  2  0  0  0    0   0    0
17  1.25  0  1.25  2  1.0   0.0  1.0   0.0
17  1.25  0  1.25  3  1.0   0.0  1.0   0.0
17  1.25  0  1.25  5  1.0   1.5708  1.0   1.5708
17  1.25  0  1.25  6  1.0   1.5708  1.0   1.5708
```

Number of disturbance forces : 4,
Fundamental frequency : F0 = 5 Hz,
Number of harmonics : 2 (5 Hz, 10 Hz),
The force f(t) is applied at the center of mass of the engine, along Ty, Tz, Ry and Rz,
$f(t)|_{Ty} = \sin(2.pi.F0.t) + \sin(2.pi.2F0.t)$
$f(t)|_{Tz} = \sin(2.pi.F0.t) + \sin(2.pi.2.F0.t)$
$f(t)|_{Ry} = \sin(2.pi.F0.t + pi) + \sin(2.pi.2.F0.t + pi)$
$f(t)|_{Rz} = \sin(2.pi.F0.t + pi) + \sin(2.pi.2.F0.t + pi)$

ACTUATOR.EXT

```
4  1
0.1 0.5
0.1 -0.5
2.4 0.5
2.4 -0.5
```

Number of actuators : 4,
Actuators are placed between the raft and the engine, near the four lower corners of the engine.
Forces are applied along the z-direction.

ERRSENS.EXT

```
4  0 0 0 0
22 0 0.6 0 3
22 0 -0.6 0 3
22 2.5 0.6 0 3
22 2.5 0 -0.6 3
```

Number of error sensors : 4,
Location : the four lower corners of the raft
Type of measurement : displacement along z direction.

MONSENS.EXT

```
0 0 0 0 0
```

Number of monitoring sensors : 0,

RESULTS

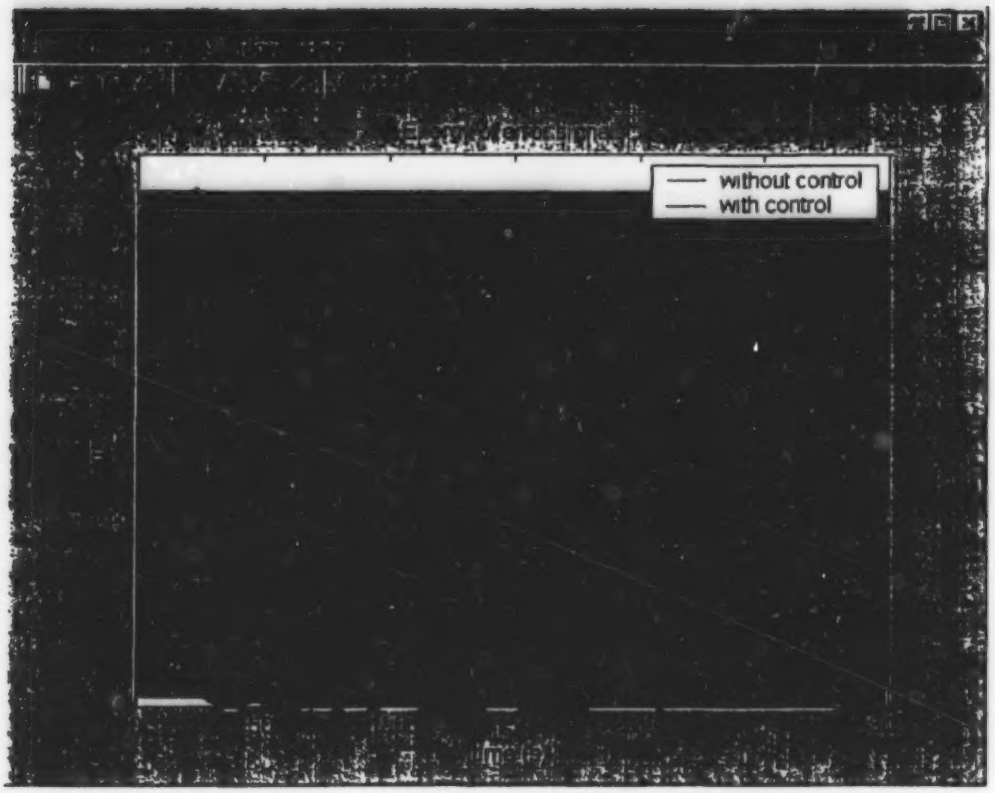


Figure A34: Energy of error signals.

Figure A34 shows the sum of the square of the four error signals. It represents the total energy of the signals measured by the error sensors placed in the four lower corners of the raft to measure the displacement in the z-direction. The active control is started at $t_0 =$ 26 s. The total energy at the error sensors is then is quicly reduced The residual signal then slowly continues to decrease, and stabilizes to a level approximately 15dB below the energy of the primary disturbance.

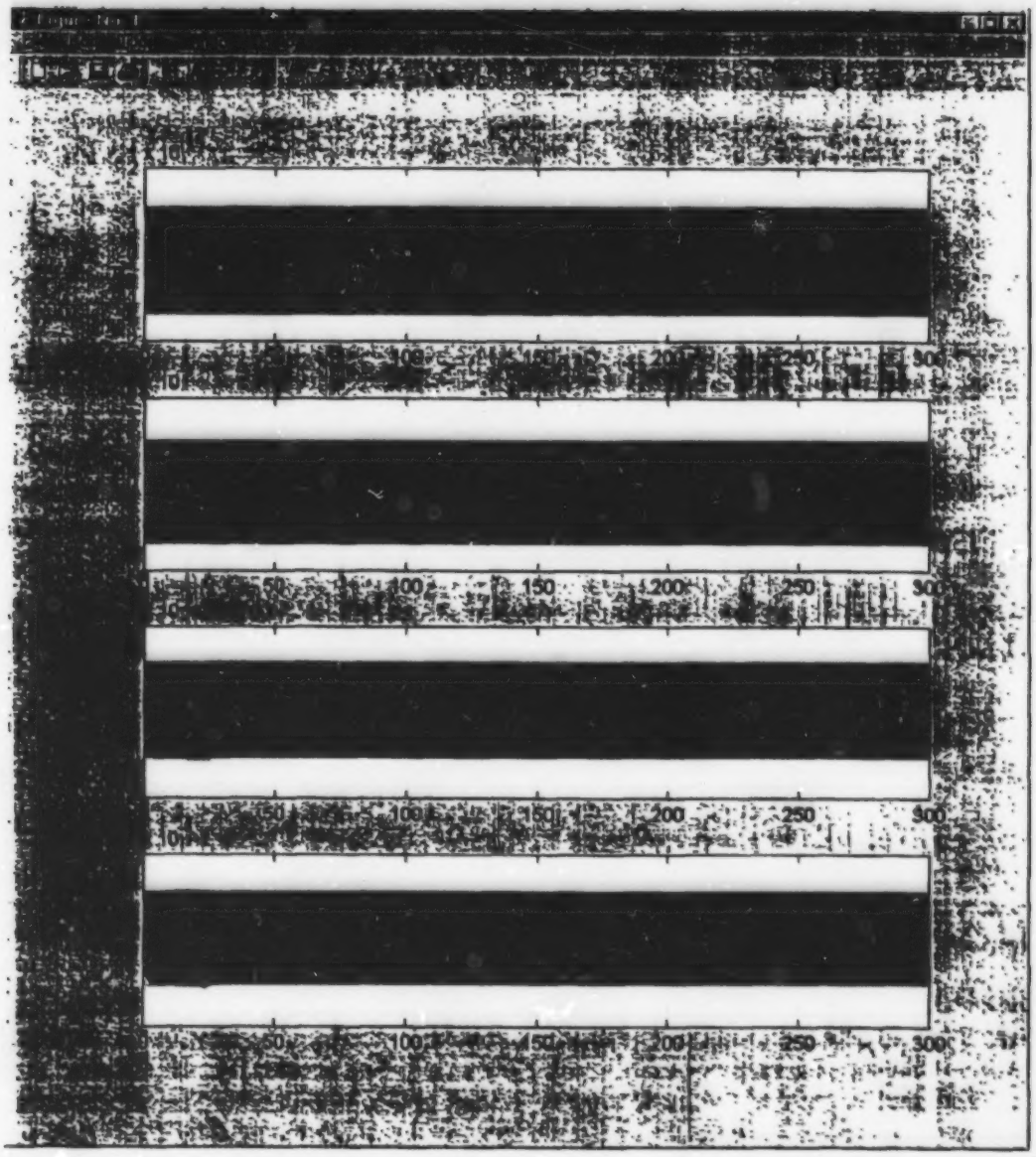


Figure A35: Individual error signals (displacement along z) as a function of time (in seconds).

The detail of each error signal (displacement along the z direction at the error sensor locations) is shown in figure A35. Because of the large coupling between actuators and sensors, we can see that after a rapid initial decrease, signals may slightly increase and decrease again. However, in spite of these oscillations, each error signal is globally decreased.

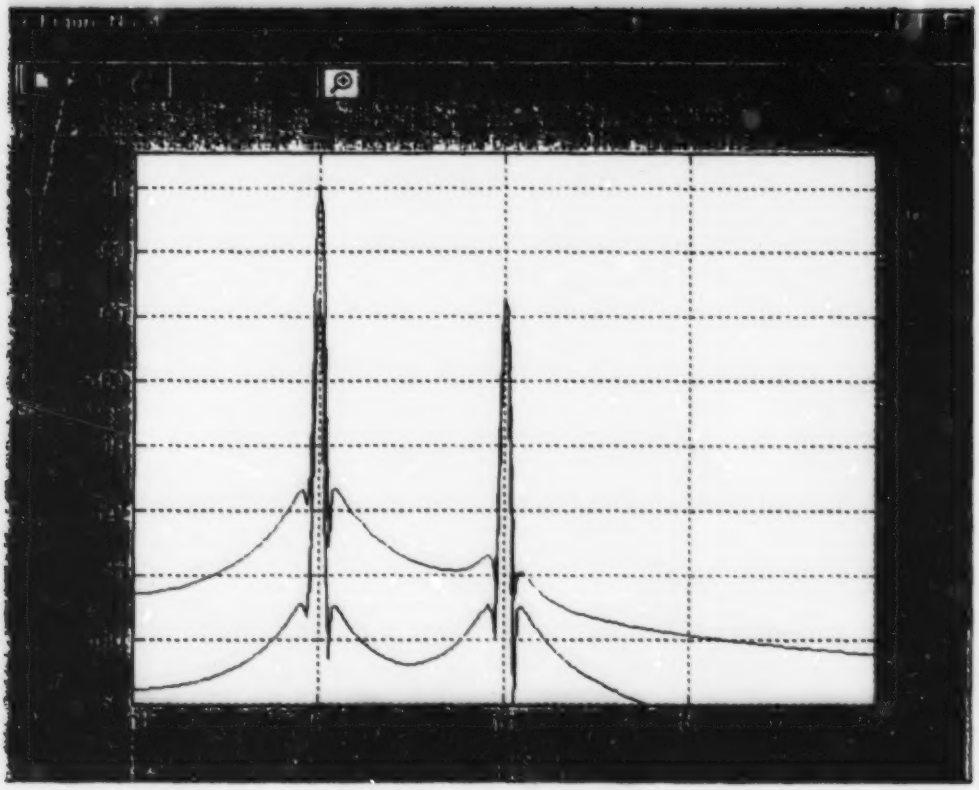
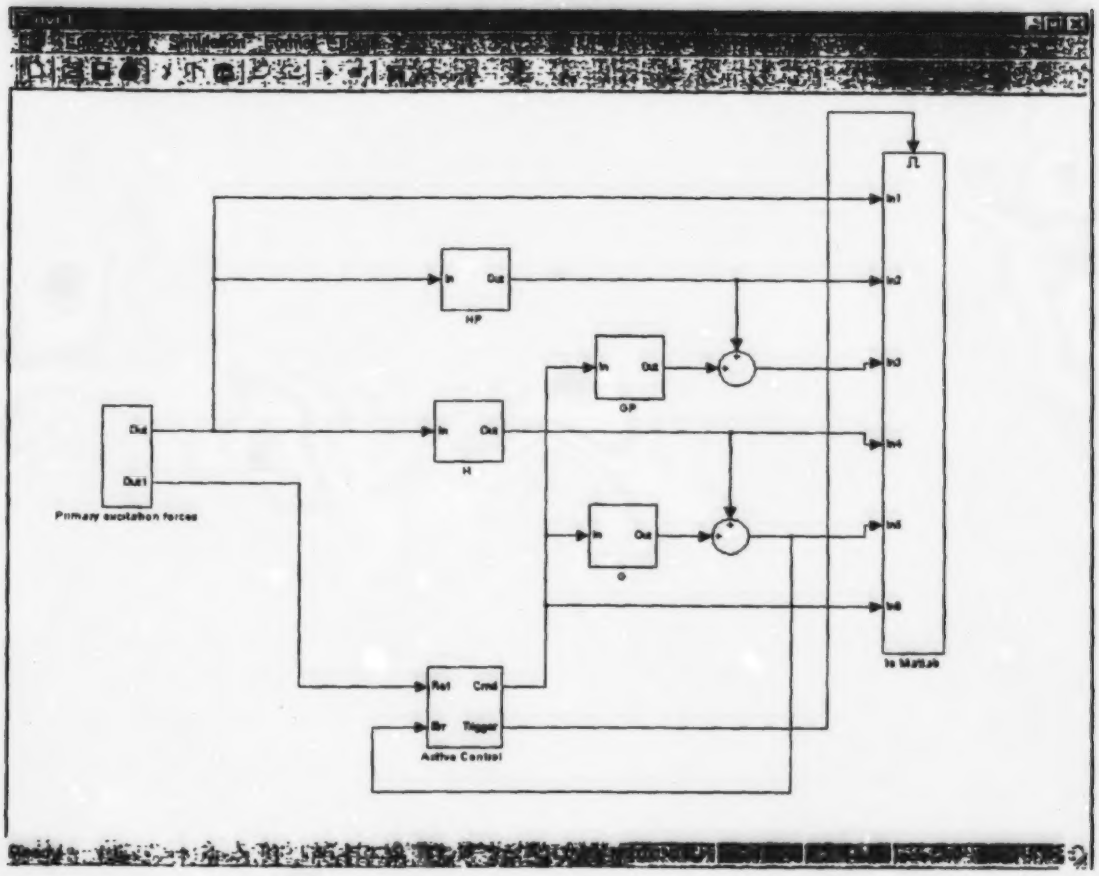


Figure A36: Error with and without control. Displacement (20 log, in meters) as a function of the frequency (Hz)

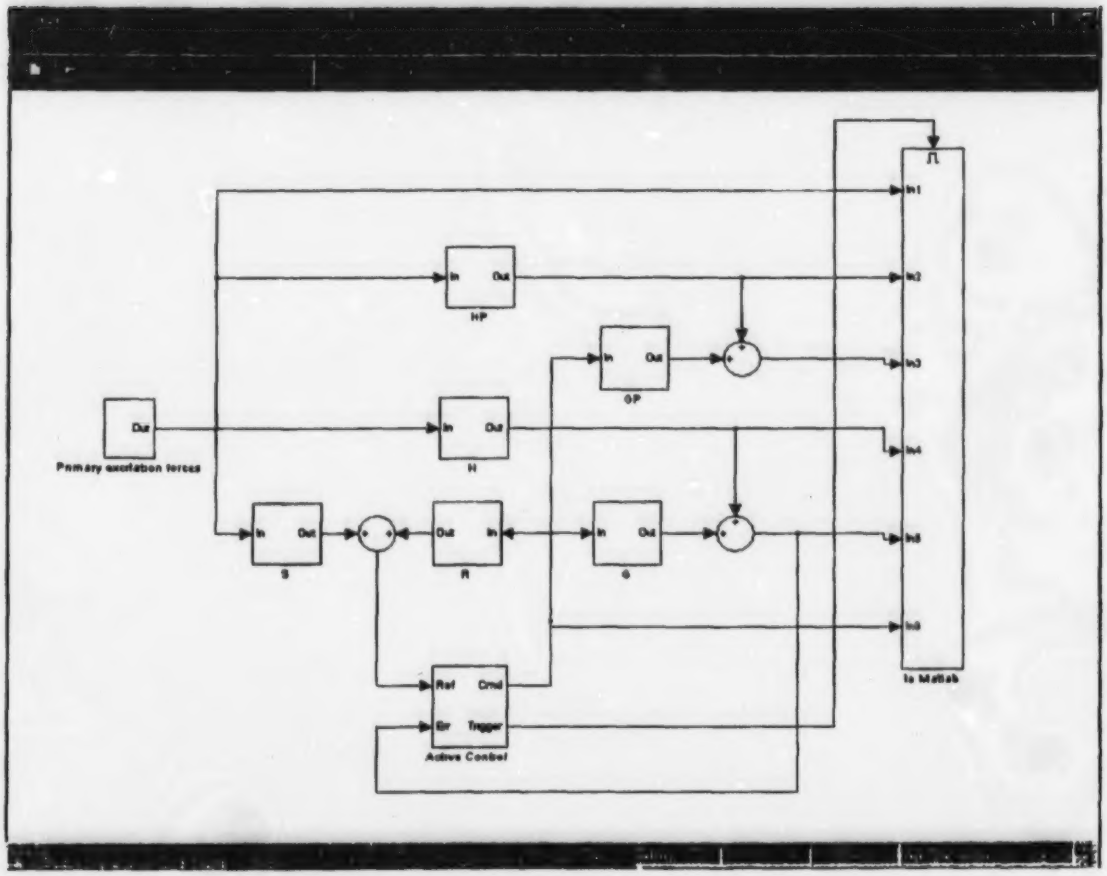
Figure A36 shows the frequency spectrum of one of the four error signals. We observe a reduction of about 15 to 20 dB at 5 Hz and about 5 dB at 10 Hz. The other spectra at the other error sensor locations (not presented her) show similar results.

VVACTIVE Appendix – Simulink Block Codes

AVC1.MDL



AVC2.MDL


In1
In2
In3
In4
In5
In6
To Matlab
HP
GP
H
G
R
Out
In
Primary excitation forces
Ref
Cmd
Err
Trigger
Active Control

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM
(highest classification of Title, Abstract, Keywords)

DOCUMENT CONTROL DATA

(Security classification of title, body of abstract and indexing annotation must be entered when the overall document is classified)

1. ORIGINATOR (the name and address of the organization preparing the document. Organizations for whom the document was prepared, e.g. Establishment sponsoring a contractor's report, or tasking agency, are entered in section 8.)

Vibe-Tek Research Ltd
5662 Holland St, Vancouver, BC, V6N 2A8

2. SECURITY CLASSIFICATION (overall security classification of the document including special warning terms if applicable).

UNCLASSIFIED

3 TITLE (the complete document title as indicated on the title page. Its classification should be indicated by the appropriate abbreviation (S,C,R or U) in parentheses after the title).

MATLAB Computer Programs for Vibration and Control of Vibrations Transmitted from Marine Engines. Users Manual

4 AUTHORS (Last name, first name, middle initial. If military, show rank, e.g. Doe, Maj John E.)

Hutton S.G

5 DATE OF PUBLICATION (month and year of publication of document)	6a. NO .OF PAGES (total containing information Include Annexes, Appendices, etc)	6b NO OF REFS (total cited in document)
January 2001	98	0

7. DESCRIPTIVE NOTES (the category of the document, e g technical report, technical note or memorandum. If appropriate, enter the type of report, e g. interim, progress, summary, annual or final. Give the inclusive dates when a specific reporting period is covered)

CONTRACTOR REPORT

8 SPONSORING ACTIVITY (the name of the department project office or laboratory sponsoring the research and development. Include address)

Defence Research Establishment Atlantic
PO Box 1012
Dartmouth, NS, Canada B2Y 3Z7

9a PROJECT OR GRANT NO. (if appropriate, the applicable research and development project or grant number under which the document was written Please specify whether project or grant).	9b CONTRACT NO. (if appropriate, the applicable number under which the document was written)
Project 1gc12, 1gh14 , and 1ga23	W7707-9-7025/001/HAL,W7707-9-7293/001/HAL

10a ORIGINATOR'S DOCUMENT NUMBER (the official document number by which the document is identified by the originating activity This number must be unique to this document.)	10b OTHER DOCUMENT NOs. (Any other numbers which may be assigned this document either by the originator or by the sponsor)
none	DREA CR 2000-078

11. DOCUMENT AVAILABILITY (any limitations on further dissemination of the document, other than those imposed by security classification)

(X) Unlimited distribution
( ) Defence departments and defence contractors; further distribution only as approved
( ) Defence departments and Canadian defence contractors; further distribution only as approved
( ) Government departments and agencies; further distribution only as approved
( ) Defence departments; further distribution only as approved
( ) Other (please specify):

12. DOCUMENT ANNOUNCEMENT (any limitation to the bibliographic announcement of this document This will normally correspond to the Document Availability (11). However, where further distribution (beyond the audience specified in (11) is possible, a wider announcement audience may be selected).

Unlimited

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM

13. ABSTRACT (a brief and factual summary of the document. It may also appear elsewhere in the body of the document itself. It is highly desirable that the abstract of classified documents be unclassified. Each paragraph of the abstract shall begin with an indication of the security classification of the information in the paragraph (unless the document itself is unclassified) represented as (S), (C), (R), or (U). It is not necessary to include here abstracts in both official languages unless the text is bilingual)

This report provides instructions for using the suite of MATLAB programs that have been developed in order to assist in the optimal vibration design of isolators for marine engines. These programs calculate the vibration transmissibility characteristics of the system and are of value in the design of optimal isolation systems. A program is also developed for the analysis of the active control of transmitted vibrations. The frequency range of interest is that encompassing the natural frequencies of the engine on its support system.

The general system that is analysed consists of an engine supported by vibration isolators on a foundation raft that itself is supported to ground by vibration isolators. The programs model the engine as a rigid body. The foundation raft can be modeled as a rigid body, or it can be modeled as a flexible body using beam finite elements. It is assumed that the isolators can be modeled as an isotropic linear viscoelastic material in terms of both a complex Young's modulus and a complex Poisson's Ratio, both of which are frequency dependant.

The report describes three different approaches to the modeling of the isolators: i) an approach based upon finite element analysis of the viscoelastic isolators using constitutive data of the isolator material; ii) an approach based upon the determination of the constitutive data from the results of dynamic stiffness measurements conducted on engine mounts; and iii) an approach based upon the results of so called 4-pole parameter data that describes the velocity force transfer matrices measured on full sized mounts in a given direction. In both the cases that involve experimentally determined isolator characteristics the data measured is not likely to completely describe the isolator characteristics. In such cases the programs developed have utilized an analytical approach to provide the missing data. In this way improved modeling of the actual structure is likely to occur. Numerical results are presented to illustrate each of the approaches developed.
A numerical problem that can occur in the finite element analysis of materials that are highly incompressible (as is the case with elastomeric materials used in vibration isolators) is described. A formulation is presented that avoids this problem. The report also contains the development of an active control option which is used to evaluate the potential for active control to be used in order to minimise the levels of force transmitted from the engine to the support.

14. KEYWORDS, DESCRIPTORS or IDENTIFIERS (technically meaningful terms or short phrases that characterize a document and could be helpful in cataloguing the document. They should be selected so that no security classification is required. Identifiers, such as equipment model designation, trade name, military project code name, geographic location may also be included. If possible keywords should be selected from a published thesaurus, e.g. Thesaurus of Engineering and Scientific Terms (TEST) and that thesaurus-identified. If it not possible to select indexing terms which are Unclassified, the classification of each should be indicated as with the title)

diesel engine vibration
vibration isolation
resilient suspension
visco elasticity
active vibration control

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM